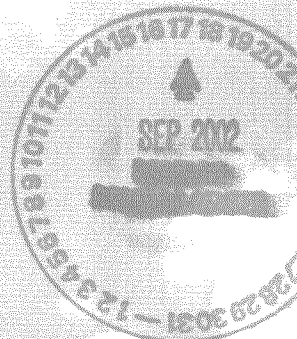
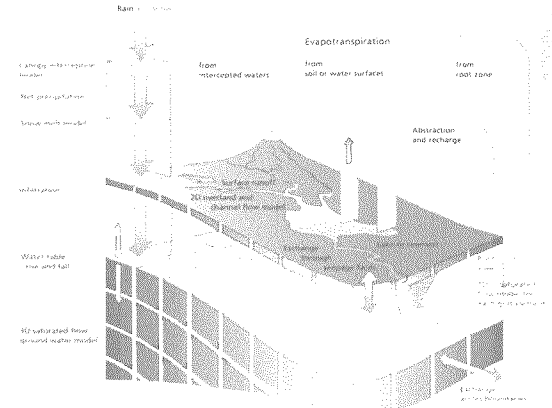




Danish Hydraulic Institute



**MIKE SHE**  
Pre- and Postprocessing

ADMIN RECORD

DOCUMENT CLASSIFICATION SW-A-004552

1/343 Best Available Copy



# **MIKE SHE PP – User Manual**

## **Overview**





## CONTENTS

1	OVERVIEW.....	1
1.1	Purpose.....	1
1.2	The User Interface - Structure and Definitions.....	2
1.3	Organising your Directories.....	5
1.4	The Different Data Files.....	6







# 1 OVERVIEW

## 1.1 Purpose

The main purpose of this manual is to enable you to apply the pre- and post-processing facilities of MIKE SHE, which you apply to manipulate and present your data. The data are typically maps of different variables (surface topography, hydraulic conductivity etc.), time series of different variables (precipitation, groundwater abstraction etc.) or results from a MIKE SHE simulation (river discharge, infiltration to the groundwater etc.).

The PP USER MANUAL is organised with a number of sections where each section deals with a specific pre-processing program.

The present manual covers the Pre- and Post-processing Package of MIKE SHE. Other MIKE SHE manuals should be read in connection with the application of other modules in MIKE SHE:

- \* MIKE SHE WM – USER MANUAL (Water Movement)
- \* MIKE SHE AD – USER MANUAL (Advection-Dispersion of Solutes)
- \* MIKE SHE WQ – USER MANUAL (Water Quality)
- \* MIKE SHE AG – USER MANUAL (Agriculture)



## 1.2 The User Interface - Structure and Definitions

MIKE SHE's user interface is based on a generalised graphical user interface which is developed by DHI using the industry standards XWindows and Motif as the basic software packages. Some facilities are general on most of the menus in the interface i.e. item selection, help facilities, file selection facilities, load and save facilities, close facilities etc. and will be described in detail here.

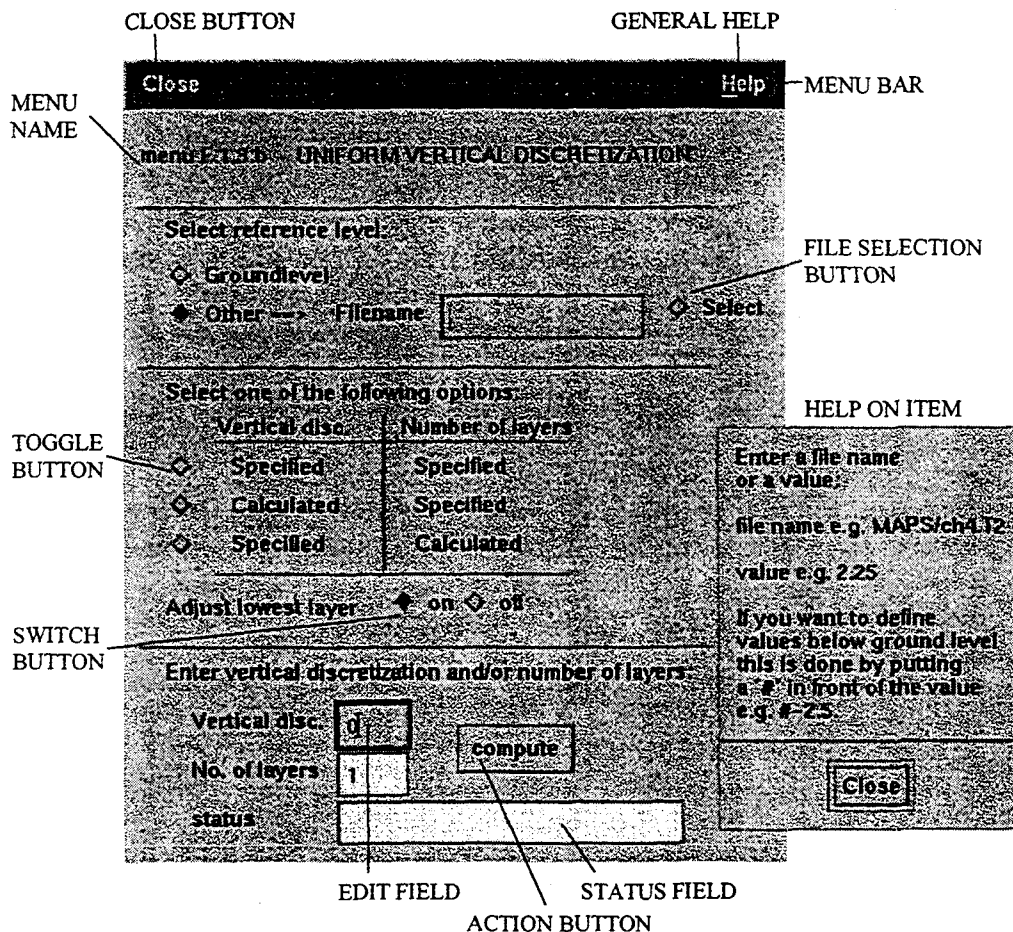


Figure 1 General facilities and expressions.

In Figure 1 some of the most used terms are shown. The list includes:

**Menubar** - the upper part of a menu which contains different options to chose e.g. <File>, <Close> and <Help>;

**Pushbuttons** - an area on the menu which activates a certain tool when the left mouse button is clicked on it;



**Toggle buttons** - buttons used to switch between two or more options or to continue to the next menu window. The button is activated also by clicking the left mouse button on it or on the text field close to it. An activated toggle button is highlighted by a red colour;

**Edit field** - an area on the menu where a value, a data file name or a text string is entered. You can change the content of an edit field by editing with the usual buttons (arrows, backspace, delete) on your keyboard. A few times of practising and you will know how it works.

If an edit field is too small to display the entire content you can also use the arrow buttons on the keyboard to display other parts of the string. On some of the edit fields you can inspect the content of an edit field by use of **arrows** on the menu which are activated by clicking the left mouse button on it. Other edit fields, item lists or data file lists have **scroll-bars** which are manipulated by holding the left mouse button down while scrolling up and down (or left and right) in the field or list;

**Pull down menus** - are menu windows from which you can select an item for a certain edit field. This kind of window is automatically closed when you have selected your item;

**Option menus** - are windows from which you can select between different options e.g. scaling: automatic / manual. This kind of window is automatically closed when you have selected the option;

### **Facilities from the menubar**

**Help** - you can get help on different levels on all menus; either as a general help by pressing the "Help" button in the upper right corner in the menubar or by clicking the right mouse button on the particular item you want help;

**File** - the <file> button is located in the menubar in the upper part of a menu window. This facility is available on each opening menu and gives the opportunity to load or save a specification file for the actual task you are about to specify. Once you have activated **File** you get a file selection window as shown in Figure 2.

You can select from a list of data files by clicking the left mouse button on the name of the data file and then press the "OK" button. Or you can specify the data file by writing the name in the "Selection" box.

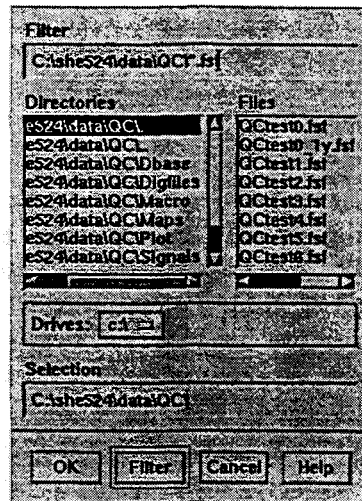


Figure 2 A file selection window.

The list of data files is chosen on basis of the specified filter given in the "Filter" box; this filter can also be modified to fit your actual choice either by clicking on a directory name and press the "Filter" button or by changing it manually. Each tool has a default filter, which will be described in the succeeding sections.

The window is automatically closed after the selection of your data file.

**Close, quit, apply, OK** - there are different ways to leave a menu; most often you just want to close the window and pass on the specifications you have given. This is done by clicking on **Close** in the menu bar - the window is closed and the specifications are in the memory of the menu program. The specifications are not saved on the specification file before you have saved with the <File> button or clicked on **OK** or **Apply** in the menu bar (this option is only present on a limited number of menu windows).

If you want to close the window without saving your specification you should click on **Quit**.

**Graphics** - on several menus it is convenient to get a graphical presentation of your input data or your results. On these menus you can activate <Graphics> from the menu bar and you will get access to the graphical presentation tool and the graphical editors in the system.

**Utilities** - in several menus it is convenient to operate on your data before entering e.g. data file names in an edit field. On these menus you can select different tools by activating the <Utilities> from the menu bar which gives you the same possibilities as <Utilities> from the opening window.



## Other facilities

**File selection window for more than one data file** - in a number of menu windows you have to specify several data file names and it is convenient to open a 'permanent' file selection window from which you can select more than one data file. This window is opened by activating the push-button <Select Files>. The search path for these data files can be set in the <File search path> edit field or in the <Filter> edit field in the selection window.

The file selection window is similar to the one you get by clicking on <File> as shown in Figure 2 and explained above. The only difference is that you have to close the window which is done by activating the <Cancel> push button;

**Cursor movements** - once you have loaded a menu window which allows you to enter some data in the edit fields you can either specify step by step and press <Return> to continue to the next edit field or you can move the mouse to the desired edit field (or an other location) and click with the left mouse button to activate this field as edit field;

The only way to get familiar with the menu system is to use it as much as possible.

## 1.3 Organising your Directories

This section describes one possible way of organising the data files for the MIKE SHE simulation package. The structure described below is used at DHI and is convenient when several persons are using MIKE SHE in various projects.

Often, several application studies are carried out simultaneously under one project and the MIKE SHE has been structured accordingly i.e. a more flexible way of defining input data files is introduced by this generation of MIKE SHE. New conventions for data file names is also introduced below.

The home directory for a given modelling application can be

/DISK/she/data/Projid/Catchid

where Projid identifies the project and Catchid indicates for example the name of the catchment you are about to develop a model for.

Under the Catchid home directory a number of sub-directories - dependent of which MIKE SHE modules you are running - should be



created. For a Water Movement and Advection-Dispersion calculation the following directories are necessary:

MAPS, DIGFILES, TIME, DBASE, PLOT, MACRO, AD, SIGNALS, tmp

The MAPS directory contains usually all the input data files of the spatially distributed data and data codes (e.g. vegetation distribution), which are retrieved and inserted in the MIKE SHE flow input file by the set-up program. Also measured data used for calibration (e.g. maps of potential heads) are stored here.

The TIME directory contains usually all the input data files of time series (e.g. precipitation), which are read by MIKE SHE during a simulation run. Also calibration series (e.g. river discharge) are stored here.

The DBASE directory contains different data bases with default parameters for some of the variables especially for calculations with the UZ component of MIKE SHE.

The DIGFILES directory contains usually digitised data of any kind. Normally, you should use the suffix 'dig' on data files with digitised data.

The PLOT directory contains all the specification files for the plots which are produced in the application. Normally, you should use the suffix 'plt' on these specification files.

The MACRO directory contains other specification files for the utility programs. The different utility programs search for data file names with different suffix - see later.

The tmp directory is used by the graphical tool to save temporary information and is the only directory which has constraints on its location and name. You should inspect the content of this directory frequently and delete old data files in order to save disk space.

Results from a simulation run with MIKE SHE are stored in two output data files a data file and a log file. They are stored at a result file directory - \$sheres - defined in your login environment file.

## **1.4 The Different Data Files**

All tools in the MIKE SHE PP system are in principle operated in equal ways. From the menu system you define a number of specifications to the tool. These specifications are written to a temporary data file. When executing the tool the specifications are read from the temporary data file. Having completed the specifications





you can write them to a specification file which you should locate on the MACRO directory. This specification file can then be loaded and modified at a later stage.

Experienced users can modify the specification file with a text editor and execute the tools "behind" the menu system.

### Specification files

Specification files have different suffix according to the task they are to be used for and they can be placed according to the directory system described in above but the user can also create his own directory structure.

The most important specification file is the one used for specifications to the MIKE SHE set-up program. It is called the **file selection file - fsf** - and has the suffix 'fsf'. The set-up program is the only task that produces a new specification file which in this case is binary. This specification file is used as the input data file for a MIKE SHE Water Movement (flow) simulation and is called the **flow input file - fif** - and has the suffix 'fif'. It is read directly by MIKE SHE and should be located on the home directory and the execution of the MIKE SHE should be made from here.

How to create the file selection file and the flow input file is described in the MIKE SHE WM User Guide.

If any errors occur during the execution of a utility tool a log-file or error file which has the same prefix as the specification file and the suffix 'err' is created and displayed automatically.

### Data files

MIKE SHE operates with three types of data files; type 0, type 2 and digitised data. They are all formatted ASCII files which can be edited by a normal text editor.

Type 0 data (time series) consists of series of individual values or scalars e.g. groundwater abstraction at different well fields. A type 0 data file should be given the suffix 'T0' in order to recognise it as time series. Each time step in the series can be different i.e. the first year data could be defined each 48 hour while the following year data are only defined each 72 hour. The precise format of time series data is defined in the **Data File Format** section in this manual.

Type 2 data (matrix data) consist of a two-dimensional array e.g. surface topography in each grid square. A type 2 data file should be



given the suffix 'T2' in order to recognise it as matrix data. The precise format for matrix data is defined in the **Data File Format** section in this manual.

Digitised data usually consists of output from the digitizer program in the Pre- and Post-processing package. Basically, this data file contains a number of x,y,z values which can be used as input to the 2D interpolation program or as landmarks in plots. See the precise format of these data files in the **Data File Format** section in this manual.

As a special data type the simulation result file from a MIKE SHE flow simulation consists of both series of matrix data and time series of a number of variables (matrix data of e.g. potential head - time series of e.g. river discharge at different locations). This data file is binary formatted and in order to inspect and present these results you can either use the graphics and operate directly on the result file (this task retrieves the specified results and presents them) or you can retrieve time series or matrix data with the output retrieval tool.



# **MIKE SHE PP – User Manual**

## **Bi-linear Interpolation (T2INTP)**





---

## CONTENTS

1	BI-LINEAR INTERPOLATION (T2INTP) .....	1
---	--	---





## 1 **BI-LINEAR INTERPOLATION (T2INTP)**

### **General Description**

With the interpolation tool (T2INTP) you are able to produce matrix data (Type 2 data files) from a limited number of point values. The tool performs an interpolation of the discrete point values in an irregular net ((x,y,z)-values) to values in a regular net ((ix,iy)-values).

### **Methodology**

The interpolation utilises sub-routines from the UNIRAS software package, using a bi-linear interpolation with up to four point values "surrounding" each output point. The values used in the interpolation are searched for within a user- specified radius. If no values are found within the search radius the output point will be assigned the delete-value (-.1e-34).





File Graphics Quit Help

menu U.6 Interpolation of discrete data (T2INTP)

Coordinate-system xbeg ybeg orientation  
0 0 0

Grid geometry nx ny dimension  
62 70 500

Search radius 2.0

Data type 60 select datatype  
Surface topography

Specification of input and output files

no.	file name(s) (input)	catchment grid file (input)
1	DIGFILES\topo.dig	MAPS\catgrid.T2 Sel.
2		
3		T2 output file name MAPS\topo.T2 Sel.
4		execute
5		status
6		

File search path  
Select files DIGFILES\*.dig

Figure 1 Menu for the T2INTP tool.

## Input data

Most often output data files from the digitising program are used as input to the interpolation program but discrete data of any kind written in a format similar to the output format of the digitising program may be used.

## Specifications

Before you start on the specifications for the interpolation program you should consider the physical location and dimensions of your matrix data.

As usual you are able to load an old specification file with by activating 'File' from the menu bar but you can also write all the specifications and save them in a new specification file.



### *Co-ordinate system:*

The definition of your co-ordinate system is used if you later want to use another co-ordinate as the basis of your set-up.

XBEG	x-co-ordinate [km] of the origin of the co-ordinate system. It can either be an UTM-system or an arbitrary user specified co-ordinate system;
YBEG	y-co-ordinate [km] of the origin;
ORIENTATION	the orientation of your matrix data [degrees (positive anti-clockwise)]; You should be very careful with different orientations of your data files!!

### *Grid geometry:*

NX	number of grids in the x-direction;
NY	number of grids in the y-direction;
DIMENSION	dimension or grid spacing [m] of your grid squares;
SEARCH RADIUS	interpolation search radius (m);

### *Data type:*

DATA TYPE	the data type is selected from the table displayed with the push button 'display data type' and is only used for identification and plotting purposes;
-----------	--

### *Specification of input and output files:*

#### DATA FILE NAME (INPUT)

You can use up till six data files containing digitised data as input to the interpolation routine. The data file names can be written directly to the edit fields but you can as usual select from a list of data files by activating the data file selection box below;



#### CATCHMENT GRID DATA FILE

If you want the interpolation to be limited to the catchment area you should specify the catchment grid code file here - otherwise let this edit field be blank;

#### DATA FILE NAME (OUTPUT)

Specify here the data file name of your output file or select from the file selection list.

The interpolation program is executed by pressing the 'Execute' field and await the message 'Normal completion' in 'Status' field below.



# **MIKE SHE PP – User Manual**

## **Polygons to Grid Data (MSHE\_OL)**





---

## CONTENTS

1	SPATIAL OVERLAY (MSHE.OL) .....	1
---	---------------------------------	---







## **1 SPATIAL OVERLAY (MSHE.OL)**

### **General Description**

With the spatial overlay tool (MSHE.OL) you are able to produce matrix data (Type 2 data files) from digitised "polygons". The tool establishes a grid code data file, which describes the spatial distribution of a parameter such as soil types, vegetation types, rainfall station network etc.

### **Methodology**

Initially MSHE.OL assigns a specified code value to all grid squares of the map. This can either be a delete value or a value, which represents the general conditions (e.g. areas with till). Internal grid squares delineated by a digitised polygon are assigned the value of the corresponding polygon code (e.g. sandy areas). If polygons surround each other the grid squares will be assigned the value of the last specified polygon.

### **Input data**

Most often an output data file from the digitising program is used as input to the overlay tool but discrete data of any kind written in a format similar to the output format of the digitising program may be used.

20



File Graphics Quit			Help		
menu U.4      Overlays (MSHE.OL)					
Coordinate-system		xbeg	ybeg	orientation	
<input type="text"/>		<input type="text"/>	<input type="text"/>	<input type="text"/>	
Grid geometry			Code values		
nx	ny	dimen.	Init. code	Add boundary	
<input type="text"/>	<input type="text"/>	<input type="text"/>	<input type="text"/>	<input type="checkbox"/> No <input checked="" type="checkbox"/> Yes	
Specification of input and output files					
file name (input)			file name (output)		
<input type="text"/>			<input type="text"/>		
<input type="text"/>			<input type="text"/>		
catchment grid file (input)					
<input type="text"/>					
Status					
<input type="text"/>			<input type="button" value="Execute"/>		

Figure 1      Menu window for the overlay tool.

## Specifications

Before you start on the specifications for the overlay tool you should consider the physical location and dimensions of your matrix data.

As usual you are able to load an old specification file by activating 'File' from the menu bar but you can also write all the specifications and save them in a new specification file.

### Co-ordinate system:

The definition of your co-ordinate system is necessary if you later want to use another co-ordinate system as the basis of your set-up.

**XBEG**                      x-co-ordinate [km] of the origin of the co-ordinate system. It can either be an UTM-system or an arbitrary user specified co-ordinate system;

**YBEG**                      y-co-ordinate [km] of the origin;



**ORIENTATION** the orientation of your matrix data [degrees (positive anti-clockwise)]; You should be very careful with different orientations of your data files!!

*Grid geometry:*

**NX** number of grids in the x-direction;

**NY** number of grids in the y-direction;

**DIMEN.** dimension or grid spacing [m] of your grid squares;

*Code values:*

**INIT. CODE** initially the overlay tool assigns all grid squares in the matrix a code value. It can either be a delete value (0 in this case) or a value which represents the general conditions;

**ADD BOUNDARY** the MSHE.OL tool is also used to establish the catchment grid code file. In this case you should press the <Yes> toggle in order to automatically add the boundary grid code (2) around the catchment area;

*Input and output data files:*

**FILE NAME (INPUT)** you can only use one data file containing digitised data as input to the overlay routine. The data file name can be written directly into the edit field but you can as usual select from a list of data files by activating the file selection box below;

**CATCHMENT GRID DATA FILE**

if you want the overlay to be limited to the catchment area (delete values outside) you should specify the catchment grid code file here - otherwise let this edit field be blank;



---

DATA FILE NAME (OUTPUT)

specify here the data file name of your output  
file or select from the file selection list.

The overlay tool is executed by pressing the 'Execute' field and await  
the message 'Normal completion' in the 'Status' field below.



# **MIKE SHE PP – User Manual**

## **Operations on Matrix Data**





## CONTENTS

1	OPERATIONS ON MATRIX DATA (T2OPR) .....	1
---	---	---

25







## 1 OPERATIONS ON MATRIX DATA (T2OPR)

### General Description

With the T2OPR tool you are able to operate on matrix data i.e. add or multiply a constant value to all your data values in one matrix data file or you can operate (add, multiply or divide) on two matrix data files. The tool is useful e.g. when you want to determine the thickness of a geological layer having the upper and lower levels of the layer or when you want to determine the transmissivity having the thickness and the hydraulic conductivity of a layer.

### Methodology

The T2OPR tool reads one or two matrix data files and determines an other matrix data file from the equation:

$$C1 + C2 * \text{file1} (+ * /) C3 * \text{file2} = \text{file3}$$

where C1, C2 and C3 are constants and file1, file2 and file3 are matrix data files. File1 and file2 should have identical dimensions.



File Graphics Quit Help	
menu U.3 Operations on matrix data (T2OPR)	
Constant 1	0.
	+
Constant 2	1.
	X
Data file 1	MAPS1topography.T2 <input type="button" value="Select"/>
	+ □
Constant 3	-1.
	X
Data file 2	MAPS1drainlevel.T2 <input type="button" value="Select"/>
	-
Output file	MAPS1draindep.T2 <input type="button" value="Select"/>
<hr/>	
Grid code file	MAPS1catgrid0.T2 <input type="button" value="Select"/>
<input type="button" value="Execute operation program"/>	
Program status:	
<input type="text"/>	

Figure 1 Menu window for operations on matrix data.

Areas with delete values (values not defined) are handled in different ways depending on the number of data files you specify and the operation you want to do:

- (1) if a 'grid code file' has been specified delete values are not allowed inside the grid code area;
- (2) if a 'grid code file' and file2 have not been specified areas with delete values in file1 will also result in areas with delete values in file3;
- (3) if a 'grid code file' has not been specified but file2 has been specified the result is as Table 3.1 indicates;



Table 1 Results of different locations of delete values in the input files.

CASE	a	b	c
file1	delete value	value	delete value
file2	value	delete value	delete value
+	$C1+C3*file2$	$C1+C2*file1$	delete value
*	C1	C1	delete value
/	C1	NOT ALLOWED	delete value

The operation can be done for the entire matrix or for a minor part of the data. In this case you have to specify a grid code file (usually the catchment grid code file) which determines the selected area.

### Input data

One or two matrix data files.

### Specifications

As usual you are able to load an old specification file by activating 'File' from the menu bar but you can also write all the specifications and save them in a new data file.

### Input and output data files

#### DATA FILE NAMES

- <Select files> gives you the opportunity to display a list of data files according to the 'File search path' and import data files from this list into the edit fields. You can also write the data file names directly into these fields. If you only want to operate on one matrix data file let the edit field for 'Data file 2' be blank;

#### CONSTANTS

- The constants in the equation is written in the edit fields according to the equation above;

#### GRID CODE FILE

- if you want the operations to be limited to the a sub-area (delete values outside) you should specify the a grid code file here - otherwise let this edit field by blank;

The operation tool is executed by pressing the 'Execute operation program' field and await the message 'Normal completion' in the 'Status' field below.





# **MIKE SHE PP – User Manual**

## **Operations on Time Series (T0OPR)**





---

## CONTENTS

1	OPERATIONS ON TIME SERIES (T0OPR) .....	1
---	---	---







## **1 OPERATIONS ON TIME SERIES (T0OPR)**

### **General Description**

With the T0OPR tool you are able to operate on time series i.e. add or multiply a constant value to all your data values in one time series or you can operate (add, multiply or divide) on two time series. The tool is useful e.g. when you want to determine the effective precipitation having precipitation and actual evapotranspiration.

### **Methodology**

The T0OPR tool reads one or two time series data files and determines an other time series from the equation:

$$C1 + C2 * \text{file1} (+ * /) C3 * \text{file2} = \text{file3}$$

where C1, C2 and C3 are constants and file1, file2 and file3 are data files with time series.



File Graphics Quit Help

menu U.7 Operations on time series (TOOPR)

Constant 1 01

+

Constant 2 11

X

Data file 1 TIMEprd.T0 Rec. no. 1 Select

+ =

Constant 3 -1

X

Data file 2 TIMEepd.T0 Rec. no. 1 Select

-

Output file TIMEnetprec.T0 Select

Output period User defined

	year	month	day	hour	min
Start date	1993	01	01	00	00
End date	1997	01	01	00	00

Execute operation program

Program status:

Figure 1 Menu window for operations on time series.

Delete values (undefined values) in either of the time series results in delete values in the new time series - except when you divide with a delete value; this is NOT ALLOWED.

## Input data

One or two time series data files.

## Specifications

As usual you are able to load an old specification file by activating 'File' from the menu bar but you can also write all the specifications and save them in a new data file.

*Input and output data files:*

## DATA FILE NAMES

- <Select files> gives you the opportunity to display a list of data files according to the 'File search path' and import data files from this list into the edit fields. You can also



write the data file names directly into these fields. If you only want to operate on one time series let the edit field for 'Data file 2' be blank;

CONSTANTS - The constants in the equation is written in the edit fields according to the equation above;

OUTPUT PERIOD - The period in which you want the operations to be performed should be specified here;

The operation tool is executed by pressing the 'Execute operation program' field and await the message 'Normal completion' in the 'Status' field below.





# **MIKE SHE PP – User Manual**

## **Retrieval of Matrix Data from Flow Input Files (MSHE.IR)**





---

## **CONTENTS**

<b>1</b>	<b>RETRIEVAL OF MATRIX DATA FROM FLOW INPUT FILES (MSHE.IR) .....</b>	<b>1</b>
----------	---	----------

35







## **1 RETRIEVAL OF MATRIX DATA FROM FLOW INPUT FILES (MSHE.IR)**

### **General description**

Input data files created by the set-up programme are binary formatted. You can retrieve data and parameters from different parts of the MIKE SHE set-up for a closer investigation. You can either retrieve a profile for a single grid square showing the distribution in depth of different parameters, a map e.g. from a single computational layer showing the spatial distribution of a single parameter or detailed information about the river network set-up.

### **Methodology**

The input retrieval program reads a Flow Input File or a Flow Result File, retrieves the specified data and writes them to an ASCII output file which you can plot, view or edit.

You can retrieve input data from the unsaturated zone (UZ) part in terms of a profile of:

- initial water content;
- conductivity related properties (saturated hydraulic conductivity, soil moisture at saturation, residual soil moisture content, and the exponent in the hydraulic conductivity function);
- retention related properties (capillary pressure at field capacity, capillary pressure at wilting point, soil moisture at effective saturation and threshold value of capillary pressure);

You can retrieve input data from the groundwater (SZ) part in terms of a profile of:

- hydrogeological parameters in combination with levels of the computational layers (horizontal and vertical hydraulic conductivity, specific storage coefficient and specific yield and initial potential head);

You can retrieve input data from the groundwater (SZ) part in terms of a map of:

- levels of the computational layers;
- horizontal and vertical hydraulic conductivity;



- specific storage coefficient and specific yield;
- initial potential head;
- boundary grid codes;
- bottom level of computational layers;
- boundary x-fluxes and y-fluxes;
- boundary x-gradients and y-gradients;

You can retrieve input data from other parts of the MIKE SHE set-up in terms of a map of:

- surface topography;
- Manning numbers and detention storage for overland flow;
- drainage levels and drainage constants;
- grid codes for the catchment boundary;
  - grid codes showing the distribution of precipitation stations, evaporation stations, temperature stations, vegetation types, soil profile types, paved areas, bypass areas, unsaturated zone calculation profiles;

### **Input data**

A Flow Input File (fif) or a Flow Result File (frf) created by the set-up program and by a MIKE SHE flow simulation, respectively.

### **Specifications**

Retrieval of input data is performed by selecting 'Input Retrieval' on the UTILITY menu; this gives you menu U.2.



File Graphics Quit Help

menu U.2 Retrieval of setup data (MSHE-IR)

Input Data File: test2.fif Select

Retrieve

◇ 1: UZ-Profile      Output Point  
◇ 2: SZ-Profile      ix:    iy:     
◆ 3: SZ-Maps      Layer number: 2  
◇ 4: Other Maps

Select Item:

Boundary grid codes  
Layer bottom levels  
Horizontal hydraulic conductivities  
Vertical hydraulic conductivities  
Unconfined storage coefficients

Output file: MAPSiszbound.T2

Execute Input Retrieval program

Program status:

View ASCII output file

Figure 1 Menu window for retrieval of set-up data.

*Select retrieval data file:*

Select your input data file either by writing the name of the fif-file or frf-file or by selecting it with the usual Select button ◇.

*Select component:*

Select the part of the MIKE SHE set-up you want to retrieve from by pressing the toggle at the desired part. The selected toggle will be highlighted.

*Select location:*

If you have selected 'UZ-Profile' or 'SZ-Profile' you must specify the grid square by it (ix,iy) value.

If you have selected 'SZ-Maps' you must specify from which computational layer the data are to be retrieved (top layer = 1).



### *Select parameter:*

Now select from the scroll-menu which parameter to retrieve by clicking the left mouse button on the desired parameter. You can scroll the menu with the scroll-bar or by clicking on the arrows in the right side of the scroll-menu.

### *Specify output data file:*

Specify the name of your output file and execute the program by pressing the 'Execute Input Retrieval Program' button.

### **Output**

The output is written in ASCII file format and you can investigate the parameters either with the 'View output' from the menu bar at the upper most part of the menu, with the graphical editor or other graphics or with a normal editor.



# **MIKE SHE PP – User Manual**

## **Retrieval of Data from Flow Results Files (MSHE.OR)**





## CONTENTS

<b>1</b>	<b>GENERAL DESCRIPTION .....</b>	<b>1</b>
<b>2</b>	<b>TECHNICAL DOCUMENTATION .....</b>	<b>2</b>
2.1	Average Value .....	2
2.2	Minimum Value .....	2
2.3	Maximum Value .....	2
2.4	Standard Deviation .....	3
2.5	Pegelweg .....	3
2.6	Exceedence Frequency .....	3
2.7	Average Duration of Exceedence Periods .....	3
<b>3</b>	<b>USER GUIDE .....</b>	<b>4</b>
3.1	General .....	4
3.2	Input Data .....	4







## 1 GENERAL DESCRIPTION

A number of new post-processing facilities have been incorporated in MIKE SHE's existing output retrieval program. The output retrieval program is used to derive results from MIKE SHE's binary result file. Derived results will either be a single time series or a T2 matrix file. These are both ASCII files and may be processed using MIKE SHE's built in data processing tools or spreadsheets, GIS systems etc.

The improved post-processing routines allow the user to generate T2 maps with statistics for any type of MIKE SHE results (ground water level, infiltration, recharge, actual evapotranspiration etc.). Thus the output will be a matrix file with statistics for each single grid. The following statistics are produced:

- Average value
- Minimum value
- Maximum value
- Standard deviation
- Pegelweg (ground water length - accumulated ground water variations)
- Exceedence frequency (Calculates the frequency where the grid value is above a certain threshold value)
- Average duration of exceedence periods (Calculates the average duration of periods where the grid values are above a user specified threshold value).
- Exceeding confidence above threshold value.

Some examples on output of the module are:

- Average ground water table, average ground water recharge or average evapotranspiration within a certain period of time.
- standard deviation on ground water fluctuations within a certain period of time
- minimum and maximum value for ground water table within a certain period of time
- Pegelweg for surface water stages within a certain period of time. Pegelweg is the sum of surface water/ground water fluctuations (m) and thus represents a measure for water level dynamics. A high Pegelweg indicates large groundwater fluctuations.



## 2 TECHNICAL DOCUMENTATION

The following statistics are produced when using the improved post-processing utility:

- Average value
- Minimum value
- Maximum value
- Standard deviation
- Pegelweg (ground water length - accumulated ground water variations)
- Exceedence frequency (Calculates the frequency where the grid value is above a certain threshold value)
- Average duration of exceedence periods (Calculates the average duration of periods where the grid values is above the threshold values)

### 2.1 Average Value

Calculating the average value for each grid.

Looping through the model domain and performing the operation

$\text{ave}(i) = \text{ave}(i) + \text{grid}(i) * 1/N$

Where N is the number of storing timestep in the specified period.

### 2.2 Minimum Value

Calculating the minimum value for each grid.

Looping through the model domain and performing the operation

$\text{mingrid}(i) = \min(\text{mingrid}(i), \text{grid}(i))$

### 2.3 Maximum Value

Calculating the maximum value for each grid.

Looping through the model domain and performing the operation

$\text{maxgrid}(i) = \max(\text{maxgrid}(i), \text{grid}(i))$



## 2.4 Standard Deviation

Calculating the standard deviation for each grid.

Looping through the model domain and performing the operation

$$\text{std}(i) = \text{std}(i) + \text{grid}(i)^2 * 1/N$$

At the end of the specified period the standard deviation is found as

$$\text{std}(i) = \text{std}(i)^{0.5} - \text{ave}(i)$$

## 2.5 Pegelweg

Calculating the Pegelweg for each grid.

The Pegelweg is given by

$$\text{Pegelweg}(i) = \text{Pegelweg}(i) + \text{abs}(\text{lastgrid}(i) - \text{grid}(i))$$

where lastgrid is the gridvalues from last storing timestep

## 2.6 Exceedence Frequency

Calculates the exceedence frequency where the grid value is above a certain threshold value. The routine finds the number of time-steps where the threshold is exceeded and divides with the total number of time-steps.

## 2.7 Average Duration of Exceedence Periods

Calculate the average duration of periods where the grid values are above the threshold values.

An exceedence period is defined as the number of hours from the point where the threshold is exceeded until the point where the value drops below the threshold again.

It should be observed that all the above statistics are calculated based on results stored in a MIKE SHE result file. These results represent average conditions over the storing time-step. Thus results should be stored with a frequency that allows a reasonable representation of the system dynamics (variations).



## 3 USER GUIDE

### 3.1 General

The improved post-processing utility is build into the existing Output Retrieval menu (menu U.1). A new option "Grid statistics" is implemented along with the selection of Grid data and Time Series data. The Grid statistics option is used to generate statistical output for any type of MIKE SHE results. The output will be a matrix file (T2).

### 3.2 Input Data

The input data are specified in menu U.1 in the MIKE SHE menu system, see Figure 1. The input data for the output retrieval is a MIKE SHE result file (frf file) and some data specifications.

**Flow Result file:** As the output retrieval works on a MIKE SHE result file (frf-file) the user will need to specify the result file to extract the statistics from.

**Nx, ny, nlay and dim:** Read only field, displaying the grid data for the selected result file.

**Simulation text:** Read only field, displaying the simulation text note for the selected result file.

**Sim. Description:** Read only field, displaying the simulation description note for the selected result file.

**Select Output type:** The user should specify whether to extract "Grid data", "Time series" or "Grid statistics". The needed input requirements will be enabled or disabled according to the selected output type. The new post-processing facilities are embedded in "Grid statistics".

**Output component:** The user should select the component from which the data should be extracted. The available types are; ET+SM (evapotranspiration and snow melt module), OC (overland and channel data), UZ (unsaturated zone data) and SZ (saturated zone data). The available data types will change according to the selected output component.

**Select data type:** Select the data type to extract data from. When a data type is selected MIKE SHE is performing a check to se if the selected



data type is stored. If the data type is not stored the user will not be able to select it.

**Start date:** The start date is by default set to the start of the selected result file. The user can change the start date by typing a new date or by using the "Step dt" function to select the time step from which the start date should be taken.

**End date:** The end date is by default set to the end of the selected result file. The user can change the end date by typing a new date. Grid statistics are calculated for the specified period.

**Output file:** The output from the output retrieval program is one or several T2 files (MIKE SHE ASCII grid files), where the user should specify the location relative to the specified MIKE SHE working directory, e.g. MAPS\testgrid.T2. When using the "Grid statistics" option the output will be eight T2 files (one for each of the statistical calculations). The following identifiers will be concatenated to the base output filename (e.g. {filename}\_MEAN.T2) Then the user specified output name will have added MEAN (average), MAX (maximum), MIN (minimum), STD (standard deviation), PGW (Pegelweg), FET (frequency above threshold), DUR (average duration of periods above threshold) and PRB (Exceeding confidence above threshold), e.g. testgridMIN.T2.



MShe

File Graphics Quit Help

menu U.1 Retrieval of simulation results (MSHE.OR)

Flow Result File

Simulation text   nx  ny  nlay  dim

Sim. description

Select output type

Output component

Select data type

- 14 : depth to phreatic surface [m]
- 15 : head elevation in saturated zone [m]
- 16 : groundwater flow components [mm/h]
- 15 : head elevation in vertical profile [m]
- 68 : Well Abstraction for Irrigation [m3/s]

Output period

Start date

year	month	day	hour	min	step dt
1998	9	6	0	0	75

End date

year	month	day	hour	min
1998	10	29	0	0

Program status:

Output file (T2-file)

Location

ix	iy	ilay	item
		1	

Threshold

abs	rel
0	0

Figure 1 Menu U.1.

**Location:** For time series the user should specify the location by a x- and a y-co-ordinate (the ix and iy box), for grid data the user should specify the calculation layer in the ilay box.

**Threshold:** The threshold values used by the grid statistics should be specified here.

The absolute threshold value is used to calculate the exceedence frequency and the average duration of exceedence periods. The relative excellence threshold allows the threshold to be defined as a fraction of the standard deviation in each grid. For instance a relative threshold value of 1.0 counts values that exceed the grid average plus a



standard deviation. Excellence frequency for the relative threshold value is written to a T2 file with the PRB concatenation.

***View Outfile:*** Displays the output file in a text editor.

The output retrieval is executed by pressing the “Execute Output retrieval” button, the progress will be shown in the “Program status” box.







# **MIKE SHE PP – User Manual**

## **Retrieval of Water Balances from Flow Result Files (MSHE.WBL)**

49





---

## CONTENTS

1	RETRIEVAL OF WATER BALANCES FROM FLOW RESULT FILES (MSHE.WBL) .....	1
---	--	---





# **1 RETRIEVAL OF WATER BALANCES FROM FLOW RESULT FILES (MSHE.WBL)**

## **General description**

The MSHE.WBL is a tool for calculating water balances for a MIKE SHE flow simulation. The MSHE.WBL calculates the water flow and storage as well as the water balance errors in the different components. The MSHE.WBL constitutes an important part of the analysis and verification of a flow simulation and should be applied whenever a flow simulation has been carried out. (In MIKE SHE Zero).

MSHE.WBL will be replaced by a new water balance program developed for MIKE SHE 1999 and onwards. A "prototype" of the new water balance program is already released with MIKE SHE 1999. This is described in the section denoted Water Balance Utility in this manual.

## **Methodology**

The MSHE.WBL reads a flow result file and calculates the water balance for the entire catchment area or for a certain part of the catchment. The water balance is calculated for a specified period of time and the output is stored on ASCII data files as time series. The output format can be a table that may be printed or viewed, or a data file of type 0 that may be plotted using MSHE.GP.

## **Input data**

The input data is a flow result file (frf) created during a MIKE SHE flow simulation. MSHE.WBL requires that water balance data is stored during the simulation.

## **Specifications**

The MSHE.WBL is accessed by selecting <Water balance> on the UTILITY menu. This activates menu U.8.



File Graphics Quit					Help								
menu U.8 Water balance calculations (MSHE.WBL)													
Flow Result File		QCtest2.frf		◇ Select		nx	ny	nlay	dim				
Simulation text		test2				31	35	1	000				
Sim. description		sz+uz+et+sm											
Select water balance type:					grid code data file								
◆ 1 : total water balance					Sub area ◇ No								
◇ 2 : water balance error					◇ Select								
◇ 3 : saturated zone (mm)					Output type: accumulated ▢								
◇ 4 : saturated zone (m3)					Output format: table (ASCII) ▢								
					Output layer: ▢								
Output period: year month day hour min					Output time step:								
start date		1981		1		1		0		0		24 (hours)	
end date		1981		2		1		0		0		1	
Execute Water balance program					Output data file: (table)								
Program status:					TIMEtotwbl.Tg								
					view outfile								

Figure 1 Menu window for the water balance calculations.

#### Selection of flow result file:

The input data file (frf) is selected by pressing the <Select> toggle button and subsequently selecting the desired data file. When the data file has been selected it is read by MSHE.WBL and the catchment geometry and simulation text and description is displayed automatically.

#### Water balance type:

The MSHE.WBL supports four different water balance types:

1. Total water balance: The water balance is calculated for all the MIKE SHE components that are included in the flow simulation.
2. Water balance error: The water balance errors are calculated for all components in the flow simulation.



3. Saturated zone (mm): The water balance is calculated for each computational layer in the saturated zone component. The output unit is mm.
4. As water balance 3, but the output unit is  $m^3$ .

#### *Sub-area water balance:*

As default the MSHE.WBL calculates the water balance for the entire model area, but a water balance may be calculated for any desired part of the model area. This is done by clicking the sub-area toggle button and subsequently specifying the name of the grid code data file that describes the sub-area. The data file must be a type 2 file (file type 21, see the Data File Format section in this manual) similar to the catchment grid code data file. It must have code value 1 inside the subarea, code value 2 on the boundary and code value 0 (or delete value) outside the subarea. Only grids having code value 1 are included in the subarea water balance. The grid data file may be prepared using MSHE.OL or the 2D-graphical editor.

#### *Output type:*

The water balance may be stored as accumulated values for the entire output period or as incremental values, giving the changes between each output time step.

#### *Output format:*

The output is stored on an ASCII file as a table or as a type 0 data file.

#### *Output layer:*

If water balance type 3 or 4 (saturated zone) is stored on a type 0 data file the output layer must be specified. The water balance for the selected layer is calculated and written to the output file. For water balance types 1 and 2 always includes the upper layer in the saturated zone and for water balances 3 and 4 all layers are included if the output is a table, hence the output layer should not be specified.

#### *Output period:*

When the flow result file is selected the output period is initialised to the entire simulation period, but any other period may be selected.





#### *Output time step:*

The output time step is initialised to the storing time step for water balance values. Using the arrow button the output time step may be changed to any multiplum of the storing time step on the flow result file.

#### *Output data file:*

Specify the name of the output file. It is recommended to use the suffix "T0" if the output file is a type 0 data file, but any file name may be specified. The <View out file> button displays the actual output file in a text-window if pressed.

### **Output data**

The MSHE.WBL calculates different important variables in the hydrological cycle. The following section explains the applied terms as well as the format of the output files. All values are average values for the entire catchment or for the specified sub-area.

#### **Water balance type 1**

The total water balance calculates the water balance for all components included in the flow simulation, however only the upper computational layer (layer 1) of the saturated zone is included, hence in order to produce a water balance for the entire saturated zone, the water balance type 3 or 4 should be applied. Water balance type 1 includes the following terms:

ac.p <sup>*)</sup>	: accumulated precipitation;
ac.ea <sup>*)</sup>	: accumulated actual evapotranspiration;
csto	: water stored in the canopy;
snow	: water stored as snow;
h-ovl	: water stored on the ground surface;
h-riv	: water stored in the river systems;
q-riv <sup>*)</sup>	: accumulated net river outflow;
thuz	: deficit in the unsaturated zone, including the unsaturated part of the upper computational layer in the saturated zone; deficit is negative;
qirr <sup>*)</sup>	: input to the catchment from irrigation;
qszb <sup>*)</sup>	: net outflow from the top layer of the saturated zone, calculated as the sum of: <ul style="list-style-type: none"><li>- flow across boundaries;</li><li>- flow to internal head boundary points;</li></ul>



- groundwater abstractions;
  - flow to the lower layers reduced with the lower layers' total exchange flow to the river, which is included in q-riv;
- qocb <sup>\*)</sup> : net outflow from overland flow across catchment boundaries;
- wblerr : The total water balance error calculated as net outflow + change in storage. The errors in the lower layers of the saturated zone are not included. Thus the error is:
- $$wblerr = -ac.p + ac.ep + q-riv - qirr + qszb + qocb + Dthuz + Dcsto + Dsnow + Dh-ovl + Dh-riv$$

where D denotes the change since last output (incremental) or changes from the start of output period (accumulated).

- <sup>\*)</sup> Accumulated from start of output period or incremental values between each output time step.

Below is shown output from a total water balance in the table output format. The type 0 data file is similar to the table format, for instance ac.p is stored as record number 1, ec.ea as record number 2 etc.

Catchment water balance (Total ,Accumulated ): c:\she524\res/QCtest2.frf

ymmmdd	ac.p	ac.ep	csto	snw	h-ovl	h-riv	q-riv	thuz	qirr	qszb	qocb	wbler
81 1 1	0	0	0	0	0	0	0	-1812	0	0	0	0
81 1 6	34	0	0	24	0	0	0	-1803	0	0	0	0
81 1 11	40	1	0	20	2	0	0	-1795	0	0	0	0
81 1 16	63	2	0	18	4	0	0	-1773	0	1	0	0
81 1 21	66	3	0	20	6	0	0	-1776	0	1	0	0
81 1 26	72	4	0	7	8	0	0	-1760	0	1	0	0
81 1 31	72	6	0	0	11	0	0	-1757	0	2	0	0

Figure 2 Total water balance (type 1). Accumulated, table output format.

## Water balance type 2

The water balance type 2 calculates the water balance error in the MIKE.SHE components that are included in the simulation. As for water balance type 1 only the top layer of the saturated zone is included. The following terms are used:

- oc.riv : error in the channel flow component, calculated as **net outflow + change in storage**;



oc.ovl : error in the overland flow component;  
uz.eps : error in the unsaturated zone, including the unsaturated part of the top layer in the saturated zone;  
wblerr : as wblerr for water balance type 1;  
other : is calculated implicit as the total error (wblerr) subtracted the errors in oc.riv, oc.ovl and uz.eps. Other errors may occur for instance if evapotranspiration or groundwater abstractions extract more water than available, from the saturated zone;

All values included in water balance type 2 are accumulated from start of output period, or between each output time step. Below is shown the table output from a MSHE.WBL water balance type 2 calculation.

Catchment water balance (ERROR ,Accumulated) : c:\she524\res/QCtest2.frf

yyymmdd	oc.riv	oc.ovl	uz.eps	other	total
81 1 1	0	0	0	0	0
81 1 6	0	0	0	0	0
81 111	0	0	0	0	0
81 116	0	0	0	0	0
81 121	0	0	0	0	0
81 126	0	0	0	0	0
81 131	0	0	0	0	0

Figure 3 Water balance error (type 2). Accumulated, table output.

### Water balance 3 and 4

The water balance for the saturated zone calculates the water budget for the computational layers in the saturated zone in mm or m<sup>3</sup>. The relevant terms are :

qszz : net flow to the layer above. For the top layer (layer 1) qszz is exchange flow to other components excluding river exchange flows and groundwater abstractions;  
qszy : net boundary outflow calculated as the sum of:  
- flow across catchment boundaries;  
- flow to internal head boundary points;  
- drainage to boundaries  
qout : groundwater abstractions;  
qszy : net exchange flow to the river;  
dszsto : change in storage in the saturated zone;  
hszmid : average potential head (m);  
wblerr : the water balance error of the actual layer calculated as net outflow from the actual layer + the change in storage.



All values, except hszmid, are accumulated from start of output period or between each output time step.

The error for layer number *i* is calculated as:

$$wblerr(i) = Dszsto(i) + qszz(i) + qszxy(i) + qout(i) + qszriv(i) - qszz(i+1)$$

qszz(i+1) is zero for the lowest computational layer.

Below is shown table output for water balance type 3, in table and type 0-output format.

a)

Catchment water balance (SZ ,Incremental ,mm ): c:\she524\res\QCtest2.frf

yymmdd	ilay	qszz	qszxy	qout	qszriv	dszsto	hszmid	wblerr
81 1 1	1	.0	.0	.0	.0	.0	42.43	.0
81 1 6	1	-4.1	.2	.0	.0	3.9	42.44	.0
81 1 11	1	-2.6	.2	.0	.0	2.3	42.45	.0
81 1 16	1	-5.3	.3	.0	.0	5.0	42.47	-.1
81 1 21	1	-3.4	.3	.0	.0	3.0	42.48	.0
81 1 26	1	-4.8	.4	.0	.0	4.4	42.49	-.1
81 1 31	1	-2.1	.4	.0	.0	1.7	42.50	.0

b)

```

FILETYPE DATATYPE VERN0:      2      51  524
TEXTLINE                :sz wbl (mm) - incremental, ilay =  1
NREC DELVAL              :    7 999.00
START DATE               :   1981  1  1  0  0
END DATE                 :   1981  2  1  0  0
1981  1  1  0  0          .0      .0      .0      .0      .0  42.4  .0
1981  1  6  0  0          4.1     -.2      .0      .0     -3.9  42.4  .0
1981  1 11  0  0          2.6     -.2      .0      .0     -2.3  42.4  .0
1981  1 16  0  0          5.3     -.3      .0      .0     -5.0  42.5  -.1
1981  1 21  0  0          3.4     -.3      .0      .0     -3.0  42.5  .0
1981  1 26  0  0          4.8     -.4      .0      .0     -4.4  42.5  -.1
1981  1 31  0  0          2.1     -.4      .0      .0     -1.7  42.5  .0

```

Figure 4 Saturated zone water balance (type 3). Incremental.  
a) table format  
b) type 0 data file format.

For plotting purposes the water balance values are calculated as positive if the flow is **into** the catchment area if the output is written to a type 0 data file.

57





# **MIKE SHE PP – User Manual**

## **Retrieval of Sub-model Head Boundary Conditions from Simulation Result Files (MSHE.BND)**

58





---

## CONTENTS

1	RETRIEVAL OF SUB-MODEL HEAD BOUNDARY CONDITIONS FROM SIMULATION RESULT FILES (MSHE.BND) .....	1
---	--	---







## **1 RETRIEVAL OF SUB-MODEL HEAD BOUNDARY CONDITIONS FROM SIMULATION RESULT FILES (MSHE.BND)**

### **General description**

For simulation of small scale hydrological or environmental problems the model set-up may not cover a well-defined hydrological unit which can be given appropriate boundary conditions. In this case the user may be forced to work on two scales with two model set-ups. For simulation with the small-scale set-up it is then possible to extract time varying head boundaries from the simulations with the large-scale set-up.

It is also possible to run a part of a large scale set-up in this way i.e. the grid sizes in the two model set-ups can be identical.

### **Methodology**

The boundary retrieval program reads the Flow Result File from a large-scale simulation, retrieves the potential head in the grids where the small-scale model requires time varying boundary data. The boundary data are then (if necessary) interpolated in time to fit the specified output time step. The boundary data are then (if necessary) interpolated in space - both horizontally and vertically - to fit the grid sizes of the small-scale model set-up.

### **Input data**

A Flow Result File (**frf**) created during a MIKE SHE flow simulation - the large scale model set-up - and a Flow Input File (**fif**) - the small-scale model set-up.

### **Specifications**

Retrieval of boundary data is performed by selecting 'Extraction of boundary conditions' on the UTILITY menu; this gives you menu U.5 (see Figure 1).



File Graphics Quit				Help			
menu U.5 Extraction of boundary conditions (MSHE.BND)							
Coarse model:							
Flow Result File	QCtest2.fri	◇ Select	mx	ny	nlay	dim	
Simulation text	test2		31	35	1	000	
Sim. description	sz+uz+et+sm						
Sub model:							
Flow Input File	test.fif	◇ Select	mx	ny	nlay	dim	
			21	21	4	100	
Output period:							
start date	year	month	day	hour	min	output time step	
	1981	1	1	0	0	24	(hours)
end date	1981	2	1	0	0		
Output File	test2.bnd	◇ Select					
Execute Boundary Extraction program							
Program status:							

Figure 1 Menu window for the tool for extraction of boundary conditions.

#### Select flow result file:

Select your input data file for the coarse model set-up, which is a result from a flow simulation, by selecting it with the usual Select button ◇. The information about simulation text, simulation description and dimensions of the model set-up will be displayed automatically.

#### Select flow input file:

Select your input data file for the fine model set-up, which is an input data file for a flow simulation, by selecting it with the usual Select button ◇. The information about dimensions of the model set-up will be displayed automatically.

#### Output period:

The output period is default the entire simulation period for the coarse model set-up but you can enter another period if desired.



### *Output time step:*

The output time step should reflect the time step in the hydrological variations i.e. you should chose small time steps if the potential head shows fast variations and vice versa.

### **Output**

Specify the output data file and execute the program is by pressing the 'Execute Boundary Extraction Program' button and await 'Normal termination' in the status window.

The output is written in binary file format and you cannot investigate the data with a normal editor.

62





# **MIKE SHE PP – User Manual**

## **Creating Time Varying Head Boundary Conditions from Observed Field Data (POD2HBD)**





## CONTENTS

1	CREATING TIME VARYING HEAD BOUNDARY CONDITIONS FROM OBSERVED FIELD DATA (POD2HBD) .....	1
---	--	---







# **1 CREATING TIME VARYING HEAD BOUNDARY CONDITIONS FROM OBSERVED FIELD DATA (POD2HBD)**

## **General Description**

POD2HBD reads a file with time series data of potential head and produces a file with time varying head boundary data, which can be used as input to MIKE SHE WM (hbd file for boundary conditions).

## **Methodology**

POD2HBD reads head data from a time series file (T0 file type 3, see the Data File Format section in this manual) and a flow input file (fif file). All saturated zone boundary grid codes containing code value 5, are identified and a head time series is attached to each of the boundary grids. POD2HBD does not interpolate in space. Thus each code 5 grid uses the nearest well (note that the T0 type 3 file contains well location). The program does not interpolate access layers. For example if you have specified code 5 to a grid in layer No. 1, the program only uses time series data for this layer (layer No. is specified in the T0 file).

No interpolation in time is carried out as well i.e. measurements are used **up to** the time specified for the measurement. If values are undefined the previous value will be used until defined values again appear. The time series should at least cover the simulation period.

The output of POD2HBD is a binary head boundary data file (hbd) file which subsequently can be used as boundary condition file for the saturated zone model.

## **Input Data**

A flow input file (fif) which has been prepared with the correct boundary codes and a time series data file with observations of potential head.

## **Specification**

The program is activated by typing pod2hbd.

65



The program prompts for the following input:

- (1) name of the data file with time series of potential head (T0 file);
- (2) flow input file of the simulation to be carried out;
- (3) output time step (hours) in hbd file;
- (4) output start date (year, month, day, hour, min.);
- (5) output end date (year, month, day, hour, min.);
- (6) output file name (hbd as file suffix is recommended).

### **Output Data**

POD2HBD produces an binary formatted data file with time varying head boundaries ready as input for a MIKE SHE WM simulation.

A file - pod2hbd.tst - with the reference system is also created and located on the directory SIGNALS. Use this file to check your set-up.



# **MIKE SHE PP – User Manual**

## **MIKE SHE 5.40 – Modflow Converter**





## CONTENTS

<b>1</b>	<b>GENERAL.....</b>	<b>1</b>
<b>2</b>	<b>TECHNICAL DOCUMENTATION.....</b>	<b>2</b>
2.1	MF2SHE.....	2
2.1.1	MODFLOW files read.....	2
2.1.2	Creation of the MIKE SHE grid.....	3
2.1.3	Topography and layer bottom.....	4
2.1.4	Hydrogeological parameters.....	5
2.1.5	Wells.....	7
2.1.6	Precipitation and evapotranspiration.....	7
2.1.7	Drains.....	7
2.1.8	River and General Head Boundary input.....	8
2.1.9	Conversion of the boundary conditions.....	8
2.2	SHE2MF.....	9
<b>3</b>	<b>USER'S GUIDE.....</b>	<b>10</b>
3.1	Input.....	10
3.2	Output.....	12
<b>4</b>	<b>REFERENCES.....</b>	<b>14</b>





## 1 GENERAL

This utility consists of two different conversion programs MF2SHE and SHE2MF which are operated from the same user interface. MF2SHE converts the input for a MODFLOW88 (McDonald and Harbaugh, 1988) model to a MIKE SHE set-up while SHE2MF converts a MIKE SHE set-up to MODFLOW input.

While MODFLOW and MIKE SHE both solve the same physical problem using the finite-difference method there are some significant differences between the two models. The main differences are:

- MODFLOW uses a variable finite-difference grid while MIKE SHE uses a square grid. This difference will imply that the conversion process will require us to transfer the spatial data from one grid system to the other. Notice that even when MODFLOW is using a square grid, data transfer is necessary when the user inputs a value for the MIKE SHE grid size different from the one used for the MODFLOW grid.
- Time dependent data are handled in MODFLOW by specifying a number of stress periods and giving input for the spatial data that changes for each of these stress periods. In MIKE SHE spatial distribution is assumed constant in time and specified in a T2-type file. The time series for each of the spatial distribution code values are stored in a separate T0-type time series file.
- In a MODFLOW model a confined aquifer can be specified using a transmissivity value and aquitards can be specified using a leakage value. In both these cases there is no need for specifying the top and bottom of aquifer and/or aquitard and system geometry is eliminated from the input data. In MIKE SHE the compartments of each model layer are characterised by a horizontal (isotropic) and vertical hydraulic conductivity and a top and bottom elevation.
- MIKE SHE is not only capable of accurately describing the groundwater flow but also contains components for modelling the unsaturated zone, overland and river flow. In MODFLOW the interaction with rivers is described as a general head boundary type sink/source.





## 2 TECHNICAL DOCUMENTATION

### 2.1 MF2SHE

#### 2.1.1 MODFLOW files read

Using the set-up name specified by the user, MF2SHE will first search for the MODFLOW files using the standard set-up name and the standard extensions used for the MODFLOW input:

- <set-up name>.BAS: the basic package file
- <set-up name>.BCF: the block centred flow package file
- <set-up name>.DRN: the drain package file
- <set-up name>.EVP: the evapotranspiration package file
- <set-up name>.RIV: the river package file
- <set-up name>.WEL: the well package file
- <set-up name>.RCH: the recharge package file
- <set-up name>.GHB: the general head boundary package file

Other MODFLOW files such as output control (OC) and solver input (SIP,SOR,PCG) will be ignored. Additional well data can be stored in files with extensions AGR (agricultural) and DOM (domestic).

The BAS file has to have the name <set-up name>.BAS. If <set-up name>.BAS is not found MF2SHE will stop and an error is written to the MF2SHE.err file. The BAS file specifies the (optional) MODFLOW packages used in the model. Other input files will first be searched using the <set-up name>.extension. If these are not found MF2SHE will look for the file using the name "fort.<unit number>" in which the unit number is the one read from the <set-up name>.BAS file. If the file is not found using this alternative name the program will write an error message to the MF2SHE.err file and stop.





converted to a 6 x 10 MIKE SHE set-up with 2 layers. The resulting boundary input for the MIKE SHE set-up is shown on the right in figure 1. The active (inside model area), boundary and inactive cells have respectively codes 1, 2 and 0. Because the extent of the active cells in layer 1 of the MODFLOW set-up is larger than for layer 2, some of the inactive cells in layer 2 will be modeled in MIKE SHE by reducing the hydraulic conductivity. These cells are indicated in grey in figure 1. Also notice that a constant head cell in the MODFLOW set-up will only be on the border in the MIKE SHE set-up if it is present in all layers.

### 2.1.3 Topography and layer bottom

In MODFLOW the model layer type (LAYCON) can be:

- 0 : confined: transmissivity and storage coefficient of the layer are constant for the entire simulation
- 1 : unconfined: transmissivity of the layer varies. It is calculated from the saturated thickness and the hydraulic conductivity. The storage coefficient is constant. This code is only valid for the top layer.
- 2 : confined/unconfined: transmissivity of the layer is constant for the entire simulation. The storage coefficient may alternate between confined and unconfined values.
- : confined/unconfined: transmissivity of the layer varies. It is calculated from the saturated thickness and the hydraulic conductivity. The storage coefficient may alternate between confined and unconfined values.

The input available in MODFLOW will depend on the layer type. Table 1. lists the information available for each layer type.



Table 1 Information available in MODFLOW according to the layer type.

MODFLOW Layer type	transmissivity	horizontal hydraulic conductivity	layer top	layer bottom	specific yield	specific storage coefficient
0	✓					✓
1		✓		✓	✓	
2	✓		✓		✓	✓
3		✓	✓	✓	✓	✓

MIKE SHE needs the topography, i.e. the top of the first model layer, and the bottom of each model layer. First the available MODFLOW data are read from the BCF file. MF2SHE then tries to find the topography and the bottom of each layer using this data. From table 1 it is clear that for the MIKE SHE topography to be available MODFLOW layer 1 has to be either type 2 or 3. If this is not the case MF2SHE will not generate the topography for the MIKE SHE set-up and a warning is written to the mf2she.log file. For the bottom of the model layers MF2SHE will try to use the top of the layer below if the MODFLOW layer type is type 0 or type 2. The top of the layer below is only available if the layer for which the bottom is needed, is not the bottom layer and the layer below has MODFLOW layer type 2 or 3. If the bottom of the layer is not known MF2SHE will write a warning to the mf2she.log file. If the topography or a layer bottom elevation is not available the corresponding entry in the MIKE SHE flow set-up file (fsf) is left blank and the user will have to complete the fsf afterwards in the MIKE SHE menu system.

#### 2.1.4 Hydrogeological parameters

The horizontal hydraulic conductivity is assumed isotropic in MIKE SHE. MODFLOW allows the user to take anisotropy into account for the horizontal hydraulic conductivity by specifying a ratio between the horizontal hydraulic conductivity in the x and y directions per layer of the grid. MF2SHE will check if the ratio is 1 for each layer and write a warning to the mf2she.log file if this is not the case. Anisotropy is ignored in the MIKE SHE set-up. The horizontal hydraulic conductivity is only directly available if the MODFLOW layer type is 1 or 2. In the other cases MF2SHE will try to convert the transmissivity using the top and bottom elevation of the layer. If this is not possible the program will store the transmissivity as a T2 file or, if



the value is constant, write the constant value to the mf2she.log file. The entry for the horizontal hydraulic conductivity is left blank in the fsf file.

MIKE SHE requires input for the vertical hydraulic conductivity (L/T) for each layer. MODFLOW however uses the leakage (1/T) between layers. This implies that for a N layer model there are N-1 leakage values in the MODFLOW input.

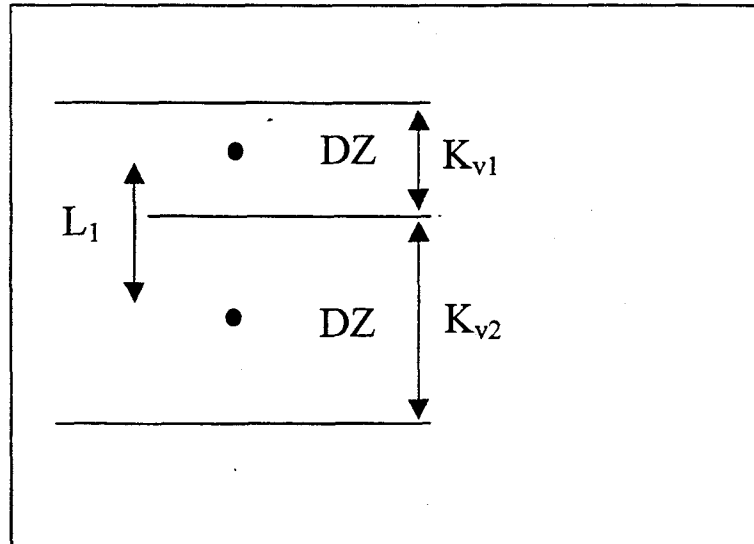


Figure 2 Relationship between the leakage (L) and the vertical hydraulic conductivity ( $K_v$ ) for a two layer setup with layer thicknesses  $DZ_1$  and  $DZ_2$ .

For the two layer set-up shown in Figure 2 the leakage between layer 1 and 2 can be calculated as:

$$L_1 = 2 \times \left( \frac{K_{v1}}{DZ_1} + \frac{K_{v2}}{DZ_2} \right)$$

where

$L_i$	: leakage between layer i and layer i + 1	(1/T)
$K_{vi}$	: vertical hydraulic conductivity for layer i	(L/T)
$DZ_i$	: thickness of layer i	(L)

MF2SHE will transfer the leakage information to the MIKE SHE grid and store it as a T2 file or, if it is a constant value, write the constant value to the mf2she.log file. In either case the entry for the vertical hydraulic conductivity is not stored in the fsf file.

If the MODFLOW set-up is a steady state set-up the values for the storage coefficients are not defined and MF2SHE will set the specific yield to 0.5 and the specific storativity to  $1.e^{-5}$  1/m and store these



constants in the fsf file. For a transient set-up, the MODFLOW layer type will determine which values are available. The specific storage in MODFLOW is dimensionless and has to be converted to specific storativity ( $1/L$ ) for MIKE SHE. This is only possible if the layer thickness can be determined from the top and bottom elevation of the layer. If the layer thickness is not available, MF2SHE will store the dimensionless value to a T2 file or, if the value is constant, write the constant value in the mf2she.log file. In either case the specific storativity is not stored in the fsf file.

### **2.1.5 Wells**

Due to changes in discretisation during the conversion process, wells which are in different grid cells in MODFLOW can be in the same grid cell in MIKE SHE. MF2SHE will therefore first aggregate wells which are in the same MIKE SHE grid cells into one well. The result is written to a T0 file and the name of this T0 file is added to the fsf file. Also notice that well abstraction is defined as negative in MODFLOW and positive in MIKE SHE.

### **2.1.6 Precipitation and evapotranspiration**

The precipitation and evapotranspiration input for MODFLOW is first converted to an effective precipitation by subtracting from the precipitation input the evapotranspiration multiplied with a factor input by the user. If the result is negative the effective precipitation is set to zero. The resulting effective precipitation is a time series of spatially distributed values. These values are first transferred to the MIKE SHE grid. If different MODFLOW grid cells map onto the same MIKE SHE grid cell only one of these cell recharge values is retained. Next MF2SHE determines the areal distribution of the effective precipitation by aggregating the input into a number of different 'precipitation stations' taking into account changes in the 'station' distribution with time by increasing the number of stations. For this distribution of stations, a time series is written to a T0 type file with one record for each of the stations.

### **2.1.7 Drains**

The MODFLOW drain input consists of conductance values and drain levels for each of the stress periods. The drain conductance values are converted to MIKE SHE drain time constants by dividing the conductance by the MODFLOW grid cell area. The time constants and drain levels can not change with time in MIKE SHE and MF2SHE will only take the distribution of the drain conductance and levels for the first stress period into account during the conversion process.



Using the spatial distribution of the drain time constants MF2SHE will determine a 'drain code map' which is used to route the drain flow in MIKE SHE. In the drain code map written by MF2HE all grid cells containing drains are set to the same code (-1) as well as the grid cells on the border of the MIKE SHE model grid. This routes the drainage flow to the model boundary. If the user wants to route the drain flow to other grid cells (e.g. containing a river) than he/she will have to edit the drain code file afterwards. During the conversion process it is possible that more than one MODFLOW grid cell maps onto the same MIKE SHE grid cell if the discretisation changes. Only one of the time constant and drain levels is retained in the resulting MIKE SHE set-up. In MIKE SHE drains can only be located in the top layer, all drains in the MODFLOW input are assumed to be located in the top layer and their input is processed accordingly.

### **2.1.8 River and General Head Boundary input**

Rivers are modeled in MODFLOW using a general head type boundary condition in which flow is only possible if the cell head for the grid cell in which the river is present, is above the river bottom. MF2SHE converts both general head boundaries and rivers to general internal head type boundaries in MIKE SHE, ignoring the river bed input for the river package input. This implies that the input for both the river and general head boundary packages is processed in the same step. During the data transfer the conductance values are rescaled to account for grid size changes. The conductances can not be time varying in MIKE SHE and only the input for the first time step will be taken into account. The head values are stored in a time series file which has to be converted afterwards using the pod2hbd utility. Also notice that general head boundary boundaries are not allowed on the model boundary in MIKE SHE. This is not a problem as the MIKE SHE model grid is extended at its edges (see 0) with extra cells.

### **2.1.9 Conversion of the boundary conditions**

In MODFLOW no-flow boundaries are not specified explicitly and occur where an active cell is next to a zero flow cell or at the edge of the model grid. In MIKE SHE on the other hand a no-flow boundary is specified by explicitly setting the cell code to zero. Internal no flow cells which occur in MODFLOW are not possible as such in MIKE SHE and are introduced by reducing the hydraulic conductivity, transmissivity and leakage of the corresponding MIKE SHE compartments to  $1.e^{-15}$  (m/s;  $m^2/s$  or 1/s). Constant head cells have a negative boundary code in MODFLOW and the corresponding grid cells in MIKE SHE are set to code 2. Grid cells with general internal head boundary conditions are set to code 8 in MIKE SHE.



If more than one MODFLOW grid cell maps to the same MIKE SHE grid cell the resulting MIKE SHE grid cell code is set according to the precedence of the codes. For active cells (inside the model boundary) this precedence is:

8 (general head boundary) > 2 (constant head) > 1 (active)

For boundary cells the precedence is:

2 (constant head) > 0 (no flow)

## 2.2 SHE2MF

The SHE2MF reads the input from a flow set-up file and converts the basic geological data to MODFLOW BAS and BCF files. As a MIKE SHE grid is always a regular grid the data can be transferred directly. Additional input for recharge, wells, drains and general head boundaries is ignored during the processing.





### 3 USER'S GUIDE

#### 3.1 Input

Input for both MF2SHE as SHE2MF is specified from the same menu (Figure 3) which can be activated from the utilities menu (Menu U).

MShe

File Quit Help

menu U.11 Modflow-MIKE SHE converter (MF2SHE/SHE2MF)

◆ MODFLOW to MIKE SHE ◆ MIKE SHE to MODFLOW

MODFLOW to MIKE SHE conversion

Setup name

MIKE SHE X-Origin  Y-Origin  Gridsize

Conversion Factors

Time MIKE SHE -  x MODFLOW

Length MIKE SHE -  x MODFLOW

Evapotranspiration

Start date year month day hour min  
1999 01 01 00 00

MIKE SHE to MODFLOW conversion

Flow Input File  ◆ Select

Execute Setup Conversion View Log file View Error file

Figure 3 Menu for the MODFLOW converter



The user chooses between a MIKE SHE to MODFLOW or a MODFLOW to MIKE SHE conversion by pressing the appropriate radio button at the top of the menu.

For a MODFLOW to MIKE SHE conversion the fields in the top part of the menu have to be filled in. The values needed are listed in Table 2.

Table 2 Input for conversion from MODFLOW to MIKE SHE.

<b>Field name</b>	<b>Meaning</b>	<b>Unit</b>
Setup name	The name used for the MODFLOW input. This name will also be used for the resulting MIKE SHE set-up.	character string (-)
X-origin	The x-co-ordinate of the bottom left corner of the bottom left grid cell in the MODFLOW grid	m
Y-origin	The y-co-ordinate of the bottom left corner of the bottom left grid cell in the MODFLOW grid	m
Gridsize	grid size to be used for the MIKE SHE grid	m
Conversion Factor / Time	Factor with which the MODFLOW time unit has multiplied to convert to hour	hour / T unit MODFLOW
Conversion Factor / Time	Factor with which the MODFLOW time unit has multiplied to convert to hour	meter / L unit MODFLOW
Conversion Factor / Evapotranspiration	Factor with which the MODFLOW evapotranspiration is multiplied before it is subtracted from the recharge during the calculation of the MIKE SHE effective recharge	(-)
Start date	Start date for the MIKE SHE simulation	year-month-day-hour-second

The MIKE SHE to MODFLOW conversion only requires the name of the flow input file (fif).



## 3.2 Output

Table 3 Possible output files from MF2SHE(<S> = set-up name; <I> = layer number). Output that is shaded requires further processing.

File name	Description
<S>.fsf	MIKE SHE flow setup file
MAPS\<S>_catgrid.T2	MIKE SHE T2 file with model boundary
MAPS\<S>_RECHARGEGRIDCO DES.T2	MIKE SHE T2 file with the precipitation distribution
MAPS\<S>_topo.T2	MIKE SHE T2 file with topography
MAPS\<S>_bottom_<I>.T2	MIKE SHE T2 file with bottom of layer <I>
MAPS\<S>_conH_<I>.T2	MIKE SHE T2 file with horizontal conductivity of layer <I>
MAPS\<S>_transmissivity_<I>.T2	MIKE SHE T2 file with transmissivity of layer <I>
MAPS\<S>_leakage_<I>.T2	MIKE SHE T2 file with leakage between layer <I> and <I+1>
MAPS\<S>_Syield_<I>.T2	MIKE SHE T2 file with specific yield of layer <I>
MAPS\<S>_Sart_<I>.T2	MIKE SHE T2 file with specific storativity of layer <I>
MAPS\<S>_Sart_dim_<I>.T2	MIKE SHE T2 file with dimensionless specific storage of layer <I>
MAPS\<S>_szgrid_<I>.T2	MIKE SHE T2 file with boundary condition codes for layer <I>
MAPS\<S>_inithead_<I>.T2	MIKE SHE T2 file with initial heads of layer <I>
MAPS\<S>_drain_codes.T2	MIKE SHE T2 file with drain codes
MAPS\<S>_drain_levels.T2	MIKE SHE T2 file with drain levels
MAPS\<S>_drain_time_const.T2	MIKE SHE T2 file with drain time constants
MAPS\<S>_SZGIHB_C<I>.T2	MIKE SHE T2 file with conductances for internal general head boundary (used to model MODFLOW rivers) for layer <I>
TIME\<S>_RECHARGE.T0	MIKE SHE T0 file with precipitation time series
TIME\<S>_abstractions.T0	MIKE SHE T0 file with abstraction data



<i>TIME</i> <S> <i>SZGIHB_Z.T0</i>	MIKE SHE T0 file with reference elevations for General Head Boundary and River packages
<i>MF2SHE.err</i>	Error and warning messages
<i>MF2SHE.log</i>	Log file detailing conversion progress and problems

MF2SHE will try to convert all available MODFLOW input. If spatial data is constant its value will be stored in the fsf file otherwise a T2 file is written and the file name is stored in the fsf file. The fsf file will only contain values or file names if the input is available after conversion as a constant or a T2 file. In all other cases the corresponding field will be empty in the fsf file. It is important to note that MF2SHE operation will never result in a complete MIKE SHE fsf file: as a minimum the user will have to convert the leakage values to vertical hydraulic conductivities. In table 3 the files which require further processing are highlighted in grey. The incomplete fsf can however be loaded into the menu system and changed appropriately after using MF2SHE. Leakage, transmissivity and the dimensionless storage coefficient can be converted to respectively vertical hydraulic conductivity, horizontal hydraulic conductivity and specific storativity after the user has determined the missing input for the topography and the elevation of model layers. The time series with the elevations of the general head boundary and river cells have to be converted to a binary <gihbd> file. The name of this binary file has already been added to the fsf file by MF2SHE. The steps for the conversion of the time series file are:

1. convert the set-up file (fsf) to a flow input file (fif)
2. use pod2hbd, a command prompt utility.

MF2SHE also writes to an error file, mf2she.err and a log file mf2she.log. These files are located in the signals directory. Mf2she.err is deleted after the file is viewed from the menu system by pressing the command button on the menu.

The SHE2MF conversion is straightforward and results in two MODFLOW files: <set-up name>.BAS and <set-up name>.BCF where <set-up name> is the same as that of the chosen fif file in the input. These result files are stored in the 'modflow' directory which is created in the working directory that contains the fif file. SHE2MF also writes a error file, she2mf.err, which details problems during program execution. She2mf.err can be found in the signals directory. Notice that this file is deleted after the file is viewed from the menu system by pressing the command button on the menu.



---

#### 4 REFERENCES

McDonald, M.G. and Harbaugh, A.W., 1988, A modular three-dimensional finite-difference ground-water flow model: U.S. Geological Survey Techniques of Water-Resources Investigations, book 6, 586 p.



# **MIKE SHE PP – User Manual**

## **Calibration Statistics Utility**





## CONTENTS

1	GENERAL DESCRIPTION .....	1
2	TECHNICAL DOCUMENTATION .....	1
3	USER'S GUIDE.....	4
3.1	General.....	4
3.2	Input .....	5
3.3	Output.....	8







## 1 GENERAL DESCRIPTION

The calibration statistic utility is a tool tailored to produce performance statistics (goodness or fit) for a MIKE SHE simulation.

The main purpose is to calculate statistics on the residuals between observed and simulated data. The current version enables the user to compare observed and simulated groundwater heads.

The present utility is developed under the project "Developing a Small Scale Integrated Surface Water and Groundwater Model for the South Florida Hydrogeologic System". The calibration utility was developed based on SFWMD traditions when calibrating a MODFLOW ground water model.

The main input is time-series files (T0-files) of observed groundwater heads. These files must include information on the location for the observation. A simulation result file including results from the saturated zone is required too.

The main output is statistical measures for each observation location and statistics on the number of locations to fulfil a number of criteria. Output can optionally be chosen as table, scattered data (.dig) or a plot macro (.plt).

## 2 TECHNICAL DOCUMENTATION

The calibration utility calculates the following basic statistical measures. Symbols and notations can be found in the list in the end of this section:

The residual or difference between observed and simulated values for each observation

$$RES_{i,j} = (H_{OBS,i,j} - H_{SIM,i,j})$$

For each location (time series) the average residual and the average of the absolute residuals



$$\overline{RES}_j = \frac{\sum RES_{i,j}}{n}$$

$$|\overline{RES}_j| = \frac{\sum |RES_{i,j}|}{n}$$

Root mean square of the residuals

$$RMS_j = \frac{\sqrt{\sum RES_{i,j}^2}}{n}$$

Standard deviation on the residuals and on the observed data

$$STD_j = \sqrt{\frac{\sum (RES_{i,j} - \overline{RES}_j)^2}{n}}$$

$$STD_{OBS,j} = \sqrt{\frac{\sum (H_{OBS,i,j} - \overline{H}_{OBS,j})^2}{n}}$$

The Nash-Sutcliffe coefficient is calculated as

$$R5_j = \frac{\sum (H_{OBS,i,j} - H_{SIM,i,j})^2}{\sum (H_{OBS,i,j} - \overline{H}_{OBS,j})^2}$$

These statistical measures are checked against different criteria. Four criteria have been implemented:

The R1 criterion ensures that the difference between residuals and the standard deviation on the residuals is kept within limits relative to the range of observed values.  $T_{R1}$  determines the acceptable range of deviation.

$$R1: ||RES_{i,j}| - STD_j| < T_{R1} (H_{OBS,max,j} - H_{OBS,min,j})$$

The R2 criterion ensures that the acceptable difference between observed and simulated values is less than the standard deviation of observations.

$$R2: H_{OBS,i,j} - STD_{OBS,j} < H_{SIM,i,j} < H_{OBS,i,j} + STD_{OBS,j}$$

The R3 criterion is applied to avoid that simulated values drop below observed minimum or exceed maximum values.



$$R3: H_{OBS,min,j} < H_{SIM,i,j} < H_{OBS,max,j}$$

The R4 criterion provides an absolute measure of the maximum allowable difference between observed and simulated values. The allowable difference,  $T_{R4}$ , is specified in meters.

$$R4: H_{OBS,i,j} - T_{R4} < H_{SIM,i,j} < H_{OBS,i,j} + T_{R4}$$

The R1-R4 criteria are checked at discrete observations within the simulation period. Often field observations and simulation results are not available at exactly the same time and normally simulation results are stored with a higher frequency. Simulated values are interpolated (linear) in time to calculate the residuals for a specific observation.

At each well location percentage of residuals to fulfil the four criteria is calculated. These percentages are checked against specified global fit criteria to determine the number of observed time series that satisfies it. Statistics on the number of observation time series satisfying the global fit criteria are determined ( $P_{R1-R4}$ ).

Definitions	Description	Unit
i	Index for time	
j	Index for location	
$H_{OBS,i,j}$	Observed value at time i at location j	m
$H_{SIM,i,j}$	Simulated value at time i at location j	m
$RES_{i,j}$	Difference between observed and simulated values	m
$ RES_{i,j} $	Absolute difference between observed and simulated values	m
n	Number of observations at one location	
N	Number of observation locations	
$STD_j$	Standard deviation of $RES_{i,j}$	m
$STD_{obs,j}$	Standard deviation of $H_{obs,i,j}$ at location j	m
$RMS_j$	Root mean square of $RES_{i,j}$	m
$H_{obs,max,j}$	Maximum value of $H_{obs,i,j}$ at location j	m
$H_{obs,min,j}$	Minimum value of $H_{obs,i,j}$ at location j	m
R1, R2, R3, R4	Numerical criteria for goodness of fit	
R5	Nash-Sutcliffe coefficient	
$T_{R1}, T_{R4}$	Tolerance constants used for criteria R1 and R4	frac., m
$P_{R1}, P_{R2}, P_{R3}, P_{R4}$	Percentage of observation where R1-R4 is meet.	%
$C_{R1}, C_{R2}, C_{R3}, C_{R4}$	Success criteria for percentages $P_{R1}-P_{R4}$	frac.

Figure 1 Definitions.



### 3 USER'S GUIDE

#### 3.1 General

In every project the calibration and verification procedures are crucial to the accuracy and reliability of the model results. The results of the calibration should be evaluated both qualitatively (visual comparison) and quantitatively (mathematically). In the beginning of a modelling study, before any modelling is done, it is useful and sound to assess the precision required by the model and to formulate calibration targets on that basis. Calibration targets can seldom be considered strict success criteria, but they should serve a quality measure. In most model calibrations some observations will not meet the targets. This is not necessary because the mathematical model is wrong. It is however the modeller's responsibility to identify a justified explanation of the problem and to suggest means to overcome the problem and to assess the importance of this on the overall model performances. The reason may be, for instance, errors in field data used for calibration or perhaps small scale features that are not accounted for by the model.

A programme for calculating statistical measures is provided – and the use of it is described in the following. The utility is aimed for projects where time-series of observed potential head constitutes a major calibration reference. As described in the technical documentation the goodness of fit of simulated potential heads are checked for a number of criteria. The criteria provides help when considering the dynamics of the ground water table with respect to both correct mean level and short and long term temporal variations.

In order to run the calibration statistics programme – the menu U.10 (Calibration Statistics) must be accessed, see Figure 2. It is a sub-menu to the general utility menu (menu U) – accessible from the main menu.



MShe

File Quit Help

menu U.10 Calibration Statistics (Calibstat)

Catchment Area

start date year month day hour min

end date

Flow Result File

Read Result File

Observations Timeseries Rec. No

Observations	Timeseries	Rec. No	
	<input type="text" value="TIMEldgu_79568.t0"/>	<input type="text" value="1"/>	<input type="button" value="Up"/> Up
<input type="button" value="Time*.T0"/>	<input type="text" value="TIMEldgu_88173.t0"/>	<input type="text" value="1"/>	<input type="button" value="Insert"/> Insert
	<input type="text" value="TIMEldgu_108152.t0"/>	<input type="text" value="1"/>	<input type="button" value="Delete"/> Delete
			<input type="button" value="Down"/> Down

Criteria R1 R2 R3 R4

Tolerance

Output type

Output file  (plt/ASCII/dlg)

Figure 2 Menu U.10 Calibration Statistics Utility.

### 3.2 Input

The basic input observed and simulated groundwater levels. Observed head must be specified in time-series files (T0-files) and simulated heads are extracted from a simulation result file. Including the information on the position of the individual observation locations together with the observed time-series (time-series files type 3, see PP Manual) enables the programme determine the corresponding location in the model – thereby making comparison with simulated heads possible.



### ***Catchment area (T2-file)***

Specify a T2 file delineating the area of interest. Calculations are performed for observation wells placed within the area. Grids included in the area should have a integer code 1 and boundary grids should have a integer code 2 - i.e. if the whole model area is of interest the catchment grid code file can be specified.

'Select' gives the opportunity to import a file from a list of files in the maps-directory. Use the file-selection box to browse to other directories if needed.

### ***Start and end date***

Specify here start and end date within which comparison between observed and simulated groundwater levels should be performed. The period must be within the simulation period - i.e. the start date should be equal or later than simulation start date and the end date must be equal to or earlier than the simulation end date.

### ***Flow Result File (frf-file)***

Specify here the name of the flow result file. The flow-simulation must include storing of the potential head in the saturated zone (datatype 15).

'Select' gives the opportunity to choose a flow result file from a list of files in the res-directory. You can only select files within the res-directory - even though using the file-selection box to browse to other directories seems to work - the calculation of the calibration statistics will not work.

### ***Read Result File***

Activating 'Read Result File' will result in retrieving simulated potential head for grid cells corresponding to where observations are measured. Retrieved data is placed in the tmp-directory.

There will be placed one file for each observation location in the tmp-directory containing the simulated time-series of the potential head. The names of these files will use the syntax - Hi\_j\_k.T0. Where i,j,k refers to the calculation cell in the saturated zone corresponding to the location of the observation well.

Retrieving data can be time consuming, and if different output type is chosen just after one another retrieving are only needed the first time. Without 'Read Result File' switched on information in the tmp-directory is used.

### ***Observation time-series and record no***

Specify here filenames of the time-series of observed groundwater levels. Indicate by the record number what time-series in the file to



use. If one file contains several time-series the file name must be repeated for each record to be used.

Only three time-series are visible – but many more can be used in the calculation of statistics. Use the 'Up' and 'Down' buttons to browse the list of specified time-series. Use the 'Insert' and 'Delete' buttons to make changes in the list. For adding time-series to the end of the list it is not necessary to use the 'Insert' – simply specify the new time-series in the empty field at the end of the list.

Time-series file names can either be typed – using a path relative to the application directory- or be selected using the 'TIME/\*.T0' button. A file-selection box will be evoked by default showing the content of the time-directory. Selecting a T0-file and pressing OK will close the file-selection box and store the filename in a buffer. Point to the field where you want the filename to be pasted and press the mouse-select button.

#### ***R1, R2, R3, R4 criteria and tolerance***

The criterion is the fraction of observations in one time-series that must satisfy the different formula (R1-R4). The observations within the specified period are checked against the R1-R4 measures and the percentage of observation to fulfil the measures is calculated for each well. These percentages are then checked against the criteria to summarise the number of observation wells where the criteria are met.

The formulations of the different measures can be found in the technical documentation section.

For the R1-criterion a tolerance constant must be specified – it is merely a multiplication number used to multiply on the difference between the maximum and the minimum head observation.

For the R4-criterion a tolerance constant must be specified. The values is used directly in the R4-measure as an absolute measure of the maximum allowable difference between observed and simulated values. Unit of the tolerance is [m].





### ***Output type***

Select here the desired output type.

Table gives a text file containing calculated criteria for each observation well. Results about R1-R4 are listed along with data of the residuals and R5 (Nash-Sutcliffe coefficient).

The different calculated data can be stored in dig files. These files contain the x and y location of the observation wells and the calculated statistical data for each well.

An option for generating a plot-macro for the graphical presentation programme is also available. Due to limitations in the plot programme the number of time series should not exceed 50.

### ***Output file***

Enter here a file name of the output data file. For dig files and plot macros it is recommended to use suffixes .dig and .plt respectively to comply with other parts of the MIKE SHE system.

### ***Execute Calibration Statistics***

Press this button to make the calculation of the calibration statistics.

### ***View Message file***

Press this button to view the message file. The message file will contain error messages if any.

### ***View Output file***

Press this button to view the result file. This is most relevant if output type is chosen to 'Table'.

## **3.3 Output**

Output in a table will result in generation of a text file. The file consists of header specifying information on which flow result file was used and some headlines. Then follows a line for each time-series specifying information about observation location and the statistical data about how well the simulation performs compared to the observations. The statistical data listed are number of observation, min. and max. observation, mean residual, RMS, STD and the percentages of observations fitted by simulated values in accordance with the R1-R4 measures and in the end is found the R5-value (Nash-Sutcliffe).

Finally statistics on how many wells fulfilled the different criteria is written. An example is found in Figure 3.





The last output option is the generation of a plot macro, which can be used in the Graphical Presentation tool. The plot macro will be constructed by placing 6 figures on each page. Due to limitations in the plotting programme this option only allows up to 50 time-series. If the plotting is activated through the Graphical Presentation menu the number of time-series should be restricted to 15 only.

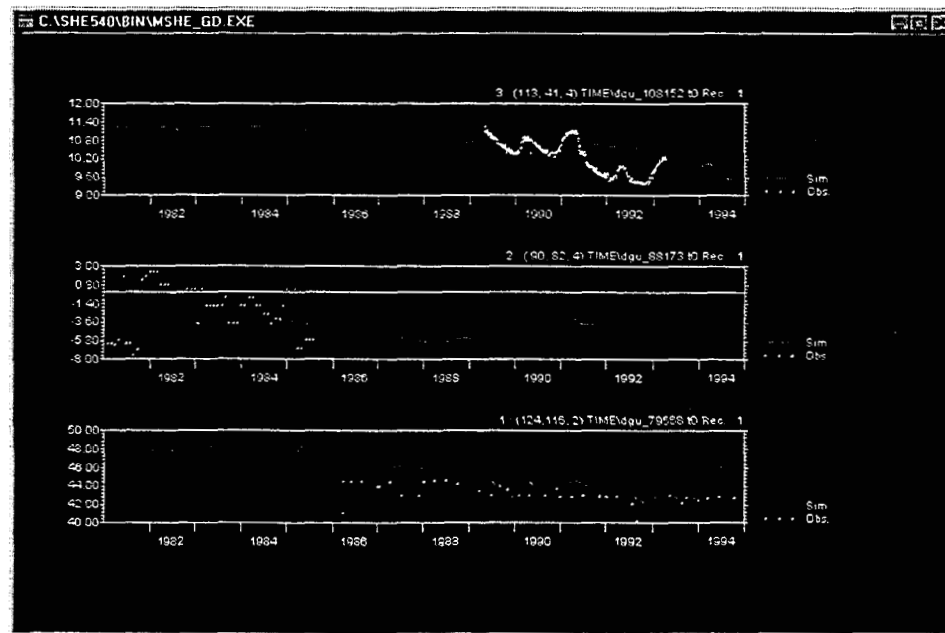


Figure 5 Plot made by using auto-generated the plot macro.



# **MIKE SHE PP – User Manual**

## **Data Aggregation Tool**

95





## CONTENTS

1	GENERAL DESCRIPTION .....	1
2	INPUT DATA.....	1
3	TECHNICAL DOCUMENTATION .....	3
4	USER GUIDE .....	4





## 1 GENERAL DESCRIPTION

The data aggregation tool produces a grid-code map (T2 file) with integer code values where each single code value refers to a certain aggregated data class which combines a number of results or input data stored in T2 files. The data aggregation tool is useful if a combination of results should be expressed in only one figure (plot). For instance in connection with ecotope mapping where a specific ecotope may be expressed in terms of a combination of hydrologic conditions. For instance a certain combination of depth to ground water table, actual evapotranspiration, flood frequency and flood duration can be represented in a single code value using the aggregation tool. The aggregated data-classes can subsequently be presented graphically in MIKE SHE's post-processor or imported to GIS.

## 2 INPUT DATA

The input data for the aggregation tool is listed in the table below

Table 1 Model Input Data.

Parameter	Description	Unit
<i>Nfiles</i>	Number of input T2 files	N/A
<i>T2_in(i)</i>	Name of T2 file number i	N/A
<i>Text_id(i)</i>	Text identifier for describing the data in T2 file (i)	N/A
<i>Ninterval</i>	Number of data intervals to be used for file (i)	N/A
<i>data_interval(i)</i>	Ninterval-1 data intervals	depends on data type in T2 files
<i>Interval_code(i)</i>	Ninterval integer code values to be assigned to interval(i)	N/A
<i>T2_out</i>	Name of T2 output file with aggregated data classes	N/A





Table 2 Model Output Data.

Parameter	Description	Unit
<i>T2_agg</i>	Aggregated data-classes stored in a T2 file with integer code values	N/A
<i>agg_key</i>	An ASCII file (key.txt) that contains a link between resulting aggregated code value and data classes in the input files.	N/A

The aggregation tool produces two different outputs. The primary output is a T2 file that contains the aggregated data classes (see Figure 1).

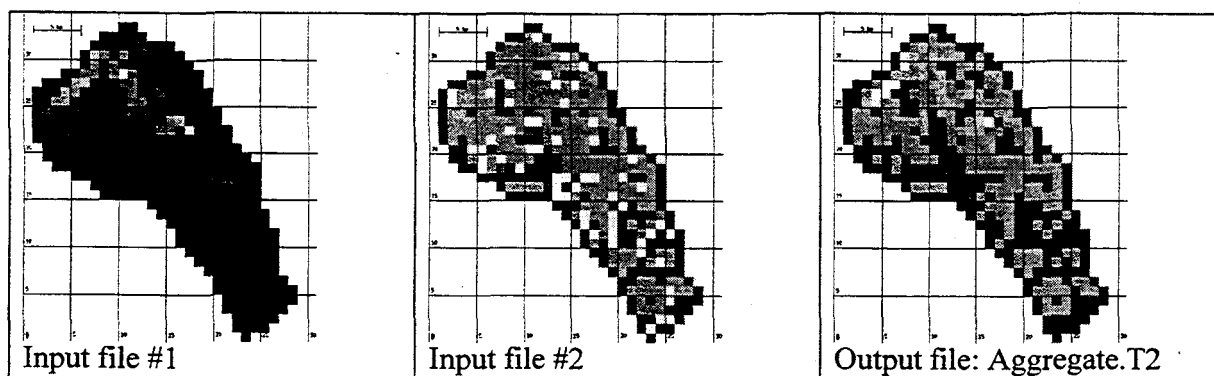


Figure 1 Aggregation of data stored in two T2 files. Output written to output file Aggregate.T2.

The second file is an ASCII file named key.txt. This file contains a link between the resulting aggregated code value (Code) and the data intervals in the input T2 files.



Aggregation key for : output.T2

Aggregate\_test\_case

ID # File type

- 1 Pegelweg (Sum of groundwater fluctuations)
- 2 Depth to groundwater table (m)

Code	File# : 1				File# : 2			
1		.0000	<= x <=	.2500		.5000	<= x <=	.6667
2		.2500	< x <=	.5000		.5000	<= x <=	.6667
3		.5000	< x <=	.7500		.5000	<= x <=	.6667
4		.7500	< x <=	1.046		.5000	<= x <=	.6667
5		.0000	<= x <=	.2500		.6667	< x <=	.7500
6		.2500	< x <=	.5000		.6667	< x <=	.7500
7		.5000	< x <=	.7500		.6667	< x <=	.7500
8		.7500	< x <=	1.046		.6667	< x <=	.7500
9		.0000	<= x <=	.2500		.7500	< x <=	1.000
10		.2500	< x <=	.5000		.7500	< x <=	1.000
11		.5000	< x <=	.7500		.7500	< x <=	1.000
12		.7500	< x <=	1.046		.7500	< x <=	1.000

### 3 TECHNICAL DOCUMENTATION

Mshe\_agg reads one or more T2 files, one or more data intervals, and generates one T2 file, which contains aggregated data. In principle the program functions as follows:

1. Read input (see Section 4. User's Guide)
2. substitute float data in the input T2 files with an integer code value in accordance with corresponding data intervals and integer code values (*interval(i)* and *interval\_code(i)*) specified as model input.
3. If more than one T2 input file is processed, each single grid point is now connected to one integer code for each input T2 file.
4. All identified combinations of integer code values are subsequently translated into one unique (aggregated) integer code value for each grid point. For instance the aggregated code value 2 may refer to the combination of code value 3 in T2 file 1, code value 1 in T2 file no. 3 etc.
5. The resulting integer grid codes are written to a T2 matrix file and an aggregation key is written to the file key.txt.



## 4 USER GUIDE

The data aggregation tool is run from a DOS-prompt (no graphical user interface is developed). The program is executed by issuing the command *mshe\_agg < input\_specification\_file*

An example of an input specification file is shown in Figure 2. Comments in *italic* refer to the input identifiers listed in Table 1 and should not be written in the input file when running the aggregation tool. The input file uses free-format.

2	<i>Nfiles</i>
FILE1.T2	<i>T2_in (name of file no. 1)</i>
average ground water table	<i>text_id (for file no. 1)</i>
2	<i>NInterval (in file no. 1)</i>
1.0	<i>data_interval (NInterval-1)</i>
1 2	<i>interval_code (NInterval codes)</i>
FILE2.T2	<i>T2_in (name of file no. 2)</i>
surface water stage	<i>text_id (for file no. 2)</i>
3	<i>NInterval (in file no. 2)</i>
0.25 0.5	<i>data_interval (NInterval-1)</i>
1 2 3	<i>interval_code (NInterval codes)</i>
aggregate.T2	<i>T2_out (aggregated code values)</i>

Figure 2 Macro file to the Data Aggregation program.



---

# **MIKE SHE PP – User Manual**

## **Graphical Viewing and Editing of Matrix Series**





## CONTENTS

1	GRAPHICAL VIEWING AND EDITING OF MATRIX SERIES.....	1
1.1	Introduction.....	1
1.2	Pulldown Menus .....	2
1.2.1	The File Pulldown Menu .....	2
1.2.2	The Coords Pulldown Menu .....	3
1.2.3	The Colours Pulldown Menu .....	4
1.2.4	The View Pulldown Menu .....	5
1.2.5	The Edit Pulldown Menu.....	7
1.2.6	The Overlay Pulldown Menu .....	8
1.2.7	The Help Pulldown Menu .....	9
1.3	Left-hand Toolbar .....	10
1.4	Top Horizontal Toolbar.....	12
1.5	Popup Menu .....	12
1.6	Free Selection .....	14
1.7	General Editing Facilities.....	15





# **1 GRAPHICAL VIEWING AND EDITING OF MATRIX SERIES**

## **1.1 Introduction**

The **SheEdit** provides the user with a number of graphical viewing and editing facilities for DHI-files of type \*.dt2, \*.dt3 and \*.T2. The dt2- and dt3-format are binary regular grid files that contain either the bathymetry or the corresponding data set. The T2-format is an ASCII-based version of dt2-files used in MIKE SHE that contains matrix data.

When **SheEdit** is started the main graphical window appears and the "Open File Selection"-dialog is posted. When a file is selected, the data is then displayed in the main window as standard map either as a box topography or a bilinear rendered map.



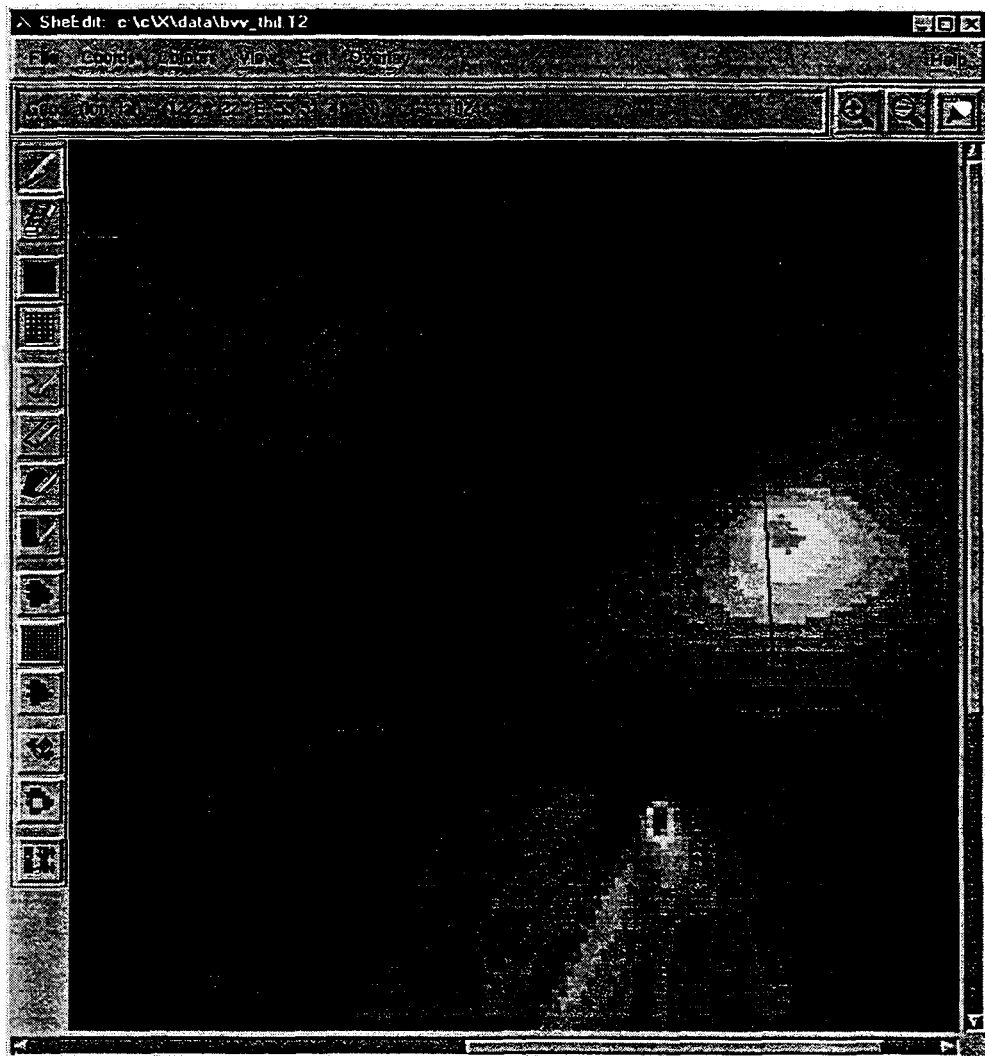


Figure 1 Topography displayed in SheEdit.

**SheEdit** features online help for all windows and dialogs, and this help is accessed by pressing the F1-key in the selected window or dialog. The general editing facilities can be accessed using the toolbar at the left-hand side of the main window or through the pulldown menus.

## 1.2 Pulldown Menus

### 1.2.1 The File Pulldown Menu

Activating the “**File**” button posts the following menu:

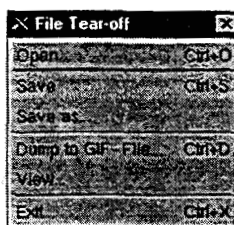


Figure 2 SheEdit's File-menu.

The entities are the following.

- Open... Activates the dialog for opening new files of various types. Available formats can be found by activating the "File Type" compobox in the "Open File Selection"-dialog. If this compobox is disabled the file format will be chosen automatically.
- Save Saves the data in the editor in the currently used data file. The user is not prompted for overwriting the file.
- Save as... Activates the dialog for saving the data in new files or in existing files. Available formats can be found by activating the "File Type" compobox in the "Save File Selection"-dialog. If this compobox is disabled the file format will be chosen automatically.
- Dump to GIF-File...  
Activates the dialog for specifying a name of the file in which the current view will be saved. When a valid file name is entered in the dialog and the "OK"-button is pressed, the current view is saved in GIF-file format.
- View... Activates the dialog for loading ASCII-text files for viewing purpose.
- Exit... Activates the "Exit"-dialog.

### 1.2.2 The Coords Pulldown Menu

Activating the "Coords" button posts a menu with the following entities:

Local Selects equidistant grid in model co-ordinates.

MIKE SHE

Selects equidistant grid in MIKE SHE model co-ordinates.  
This option is default.



**Geo** Selects grid in geographical co-ordinates. This function is not available unless the data file contains valid information about geographical location and orientation.

**UTM, SBF, DKS, OSGB, BTM, HKG, GAUSS-BOAGA, AMG, GAUSS-KRUGER, T.S.CASSINI**

Selects grid in UTM (or in the chosen local UTM-like) co-ordinates. This function is not available unless the data file contains valid information about geographical location and orientation.

**None** Deselects grid.

**Edit UTM Zone...**

Activates the dialog for selecting UTM zone. The user can choose either autocalculation of the UTM zone, which is based on the centre of the map, or a specific fixed UTM zone. The dialog is only available when UTM co-ordinates are selected.

**Layout...** Activates the dialog for selecting direction of the mod-co-ordinate axis. This function is only available when network files are loaded.

### **1.2.3 The Colours Pulldown Menu**

Activating the "**Colours**" button posts a menu with the following entities:

**Red - Green - Blue**

Selects a linear autoscaled palette consisting of the colours red, green, blue and intermediate colours. The palette consists of 20 colours. This palette is default.

**Black - Red - Yellow**

Selects a linear autoscaled palette consisting of the colours black, red, yellow and intermediate colours. The palette consists of 20 colours.

**Wheat - Green - Blue**

Selects a linear autoscaled palette consisting of the colours wheat, green, blue and intermediate colours. The palette consists of 20 colours.



#### Lightblue - Darkblue

Selects a linear autoscaled palette consisting of the colours from lightblue to darkblue. The palette consists of 20 colours.

#### White - Black

Selects a linear autoscaled palette consisting of the colours from white to black. The palette consists of 20 colours.

#### White - Grey

Selects a linear autoscaled palette consisting of the colours from white to grey. The palette consists of 20 colours.

#### Land Water 1

Selects a two-colour land water palette, one colour for land and one for water.

#### Land Water 2

Selects a land water palette, linear autoscaled in depth only. The palette consists of a special land colour and 10 blue colours for the water.

#### Edit RGB...

Activates the dialog for colour editing. The user is prompted for an index in the colour table and the "Single RGB Select"-dialog is popped up. Note that if either the "Land Water 1"- or the "Land Water 2"-palette is selected the land colour cannot be edited in this dialog.

#### Land Colour..., Grid Colour..., Delete Colour...

Activates the dialog for selecting land/grid/delete colour. The user can choose between 20 predefined non-editable colours.

#### Colours...

Activates the advanced colour control dialog. The user can zoom in depth, choose between scaling types, edit and select delete values, load, save and create new palettes.

### 1.2.4 The View Pulldown Menu

Activating the "View" button posts a menu with the following entities:

**Coords** This button toggles monitoring of the co-ordinates for the mouse pointer on and off. When switched on the mouse pointer position is displayed in the text field right below the menubar. The co-ordinates are displayed in the same co-



ordinate system as selected in the "Coords"-menu. For dt2-, dt3- and T2-files the height z is displayed as well. The button is default switched off.

#### Bilinear Interpolation

This button toggles bilinear interpolation between the grid points on and off. When switched on the topography is rendered as shaded isolines. The button is default switched off.

#### Fixed Scale

This button toggles displaying of data scaled according to global minimum and maximum on and off. The button is only available when time series for dt2- or dt3-files are displayed. The button is default switched off.

#### Overview...

Activates the window for displaying the overview (the entire data set).

#### 3D...

Activates the window for displaying the current view in 3 dimensions.

#### Scale...

Activates the window for displaying the colours and the related values. When double clicking on one of the colours the "Single RGB Select"-dialog for editing the RGB-values is popped up with the current colour. Note that the land, if present, cannot be edited.

#### Zoom In

Activates the zoom in function. The user then has to click, drag and release the mouse pointer to select the desired area as the new view.

#### Zoom Out

Activates the zoom out function, that zooms all the way out.

#### Previous

Recalls the previous zoom.

#### Refresh

Activates the function for redrawing the entire topography as well as overlays.

#### Select Point

Activates the select point function. The user moves the mouse pointer to the desired location and clicks in the map. This is used when the "3x3 Mask" function is active.



### 1.2.6 The Overlay Pulldown Menu

Activating the “**Overlay**” button posts a menu with the following entities:

**Values** This button toggles display of values for each grid point on and off. The values are displayed with 3 significant digits or in the shortest possible way in the centre of the grid point. If the number cannot be displayed in neither way 2 significant digits in the number is tried. The values are only displayed if there is space for it. This has the effect that with certain grid sizes the values will be displayed scattered. If the grid size is too small, no values will be displayed. The button is default switched off.

**No XYZ** Selects no xyz-data displayed as overlay. The button is default switched on.

**XYZ** Selects xyz-data displayed on top of the topography. Each single digitised xyz-point is drawn using the same colour scale as the topography it self. This enables the user to see if the grid points and the digitised xyz-points are matching correctly. The button is default switched off.

**XYZ + Frame**

Selects xyz-data displayed on top of the topography. Each single digitised xyz-point is drawn using the same colour scale as the topography it self. In order to see the point if it is drawn on top of a grid point with the same colour each point is surrounded by black frame. The button is default switched off.

**XYZ + Frame + Value**

Selects xyz-data displayed on top of the topography. Each single digitised xyz-point is drawn using the same colour scale as the topography it self. In order to see the point if it is drawn on top of a grid point with the same colour each point is surrounded by black frame. At the right side of the frame is the actual value displayed. The button is default switched off.

**Contour** Selects xyz-data displayed on top of the topography. The xyz-data is displayed as open contours. Each contour is displayed as a black polyline, and the height of the contour is labelled at the end of the polyline. If the xyz-data has no



### Select Area

Activates the select area function. The user then has to click, drag and release the mouse pointer to select the desired area. In this rectangular area all the grid points are selected. This function duplicates the "Select Rectangle" function.

## 1.2.5 The Edit Pulldown Menu

Activating the "Edit" button posts a menu with the following entities:

### 3x3 Mask...

Activates the "3x3 Mask"-dialog in empty form. The user can then select a single grid point as the centre of the "3x3 Mask" and then freely edit in the data.

### Operations...

Activates the "Operations"-dialog. This dialog contains filters for selecting grid points and operations to perform with the selected data.

**Filters...** Activates the "Filters"-dialog. The dialog contains 9 different filters for smoothing, sharpening and deriving the data. These functions can operate on the entire data set of the selected data.

### Statistics...

Activates the "Statistics"-window. This dialog displays statistical values and the distribution function for the selected grid points. Information about single intervals can be obtained by clicking with the right hand mouse button on one of the columns in the plot of the distribution.

### Resample...

Activates the dialog for resampling the topography. The "Resample"-dialog can only be activated when topographies have been loaded. The dialog is unavailable when dt2- or dt3-timeseries have been loaded.

### Cut...

Activates the dialog for cutting out a subarea in the topography. The "Cut Data"-dialog can only be activated when topographies have been loaded. The dialog is unavailable when dt2- or dt3-timeseries have been loaded.



contour information nothing will be drawn. The button is default switched off.

#### Import XYZ...

Activates the dialog for loading digitised data (xyz-data - \*.xyz). This enables the user to check if the original digitised data matches with the topography. Note that the correct co-ordinate system for the digitised data has to be specified. The same type of co-ordinates as in the "Coords"-menu are supported. The xyz-points are then converted to model co-ordinates and displayed on top of the topography if either "XYZ", "XYZ + Frame" or "XYZ + Frame + Value" is selected. Loading of this file appends to already loaded points.

#### Export XYZ...

Activates the dialog for saving currently loaded digitised data in xyz-format. Every digitised point will be mapped from the actual xy-values to the chosen type of co-ordinates. Note that the same type of co-ordinates as in the "Coords"-menu are supported.

#### Import DIG to XYZ...

Activates the dialog for loading digitised data in MIKE SHE format (dig-data - \*.dig). This enables the user to check if the original digitised data matches with the topography. The co-ordinates for the data are automatically interpreted as MIKE SHE model co-ordinates. Selecting either "XYZ", "XYZ + Frame" or "XYZ + Frame + Value" displays the co-ordinates on top of the topography. Loading of this file appends to already loaded points.

#### Delete XYZ (# points)

Deletes all loaded xyz-points.

### 1.2.7 The Help Pulldown Menu

Activating the "**Help**" button posts a menu with the following entities:

General... Activates the "Help Text"-dialog. In general help can be obtained by pressing F1. The help system then searches recursively back until it finds an object with help attached. The help text is then displayed in the "Help Text"-dialog.

#### About SheEdit...

Activates the "About SheEdit"-dialog.





### 1.3 Left-hand Toolbar

The toolbar consists of a number of small icon buttons. Each of them activates functions that are used for selecting data. When the mouse cursor is moved inside a bitmap button a small flyby help displays the function of the button. This is used as the text in the following list of functions in the toolbar.



Select

When the functions "Select/Unselect Points", "Select/Unselect Line Segments", "Select/Unselect Polygon" and "Select/Unselect Rectangle" are chosen this option ensures that these four functions selects grid points. This option is default switched on and complements "Unselect".



Unselect

When the functions "Select/Unselect Points", "Select/Unselect Line Segments", "Select/Unselect Polygon" and "Select/Unselect Rectangle" are chosen this option ensures that these four functions deselects grid points. This option is default switched off and complements "Select".



Shade

This button controls how the selected grid points are displayed. When chosen the selected grid points are shaded out with the currently chosen grid colour. This option is default switched off and complements "Frame".



Frame

This button controls how the selected grid points are displayed. When chosen the selected grid points are indicated by a surrounding frame coloured with the currently selected grid colour. This option is default switched on and complements "Shade".



Select/Unselect Points

This button activates the function for freehand selection/unselection of grid points. Selection/unselection is determined by the selection state ("Select" or "Unselect").



#### Select/Unselect Line Segments

This button activates the function for selecting/unselecting multiple connected line segments. Selection/unselection is determined by the selection state ("Select" or "Unselect").



#### Select/Unselect Polygon

This button activates the function for selecting/unselecting a filled polygon. Selection/unselection is determined by the selection state ("Select" or "Unselect").



#### Select/Unselect Rectangle

This button activates the function for selecting/unselecting a filled rectangle. Selection/unselection is determined by the selection state ("Select" or "Unselect").



#### Select Values

This button activates the function that automatically selects all the grid points where the value is not equal to the delete value.



#### Select All

This button activates the function that selects all the grid points in the topography, actual values as well as delete values.



#### Select View

This button activates the function that selects all the grid points in the current view of the topography, actual values as well as delete values. When zoomed out the function does the same as the "Select All" function.



**Clear All** This button activates the function that deselects all the selected grid points in the entire topography.



#### Clear View

This button activates the function that deselects all the selected grid points in the current view. When zoomed out the function does the same as the "Clear All" function.



#### Min/Max Scale

This button activates the function for rescaling the topography to the minimum and maximum of the data.

## 1.4 Top Horizontal Toolbar

The top horizontal toolbar contains a message field for displaying various messages and monitoring of co-ordinates for the mouse cursor. Besides this message field it contains the following three buttons:



**Zoom In** This button activates the zoom in function. The user then has to click, drag and release the mouse pointer to select the desired area as the new view.



**Zoom Out** This button activates the zoom out function that zooms all the way out.



**Refresh** This button activates the function for redrawing the entire topography as well as overlays.

## 1.5 Popup Menu

Clicking with the right hand mouse button in the main map posts a popup menu containing the following entries:

**Zoom In** Activates the zoom in function. The user then has to click, drag and release the mouse pointer to select the desired area as the new view.

**Zoom Out**  
Activates the zoom out function, that zooms all the way out.



**Previous** Recalls the previous zoom.

**Refresh** Activates the function for redrawing the entire topography as well as overlays.

**Overview...**

Activates the window for displaying the overview (the entire data set) which includes options for changing the area by zooming or panning. The "Overview"-window pops up with the view centred around the cursor and is automatically destroyed when the cursor is moved outside the window.

**Select Point**

Activates the select point function. The user moves the mouse pointer to the desired location and clicks in the map. This is used when the "3x3 Mask" function is active.

**Select Area**

Activates the select area function. The user then has to click, drag and release the mouse pointer to select the desired area. In this rectangular area all the grid points are selected. This function duplicates the "Select Rectangle" function.



## 1.6 Free Selection

Double clicking with the left-hand mouse button in the main map automatically selects a grid point. If the "3x3 Mask" is active the selected grid point is displayed for editing purpose in this window. In the "3x3 Mask" is inactive, it is posted with the selected grid point. This is illustrated on the following figure:

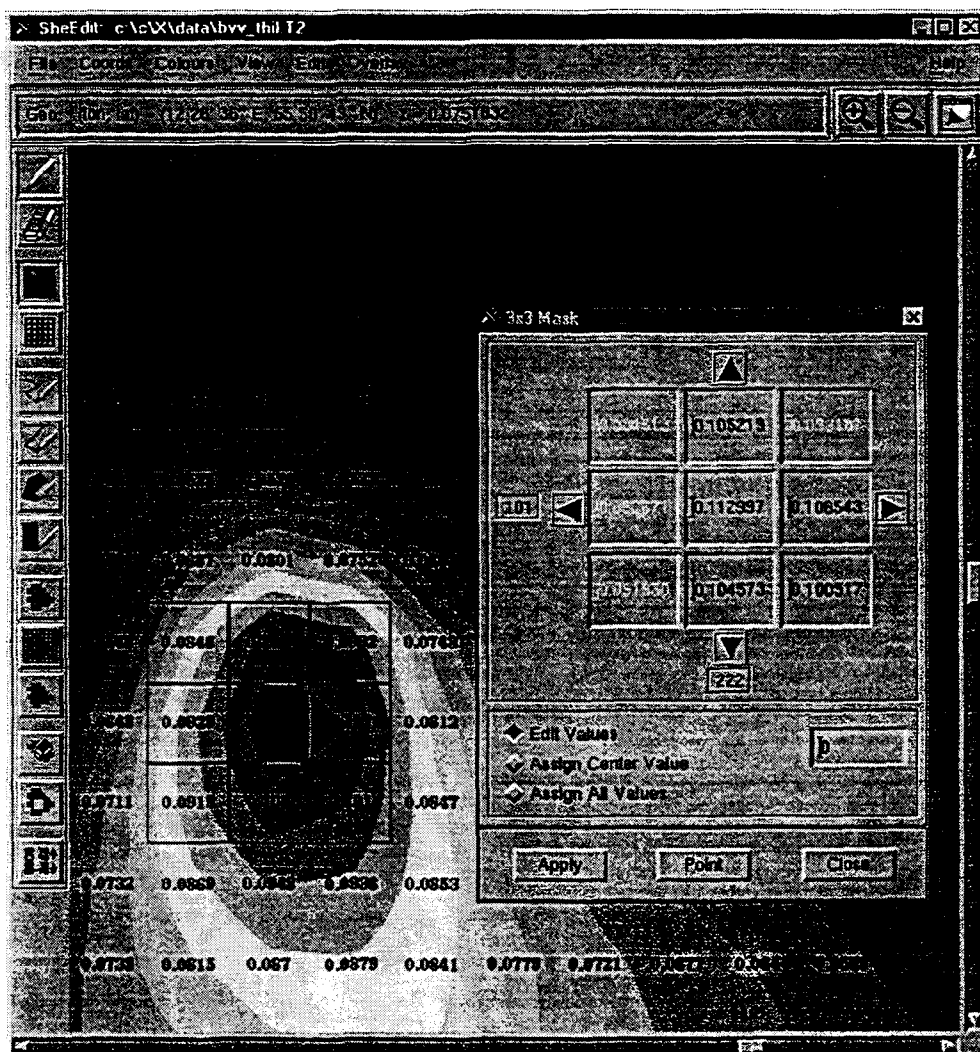


Figure 3 Bilinear rendered map and 3x3 Mask.

The "3x3 Mask"-dialog displays the levels in the bathymetry. Note that some numbers are yellow (bright) and others are black. The bright number indicates that the text field is too short to display the entire number. Clicking with the right hand mouse button in the field expands it, so the field accommodates to the length of the number.



## 1.7 General Editing Facilities

**SheEdit** provides the user with the possibility for viewing the data as a surface in 3 dimensions. The user can control the position of the viewpoint and thereby view the data from different angles. The window also includes functions for scaling the height of the surface, viewing the data as a solid model or a wireframe and colouring the surface. The following figure shows a bathymetry displayed in the “3d View”-window.

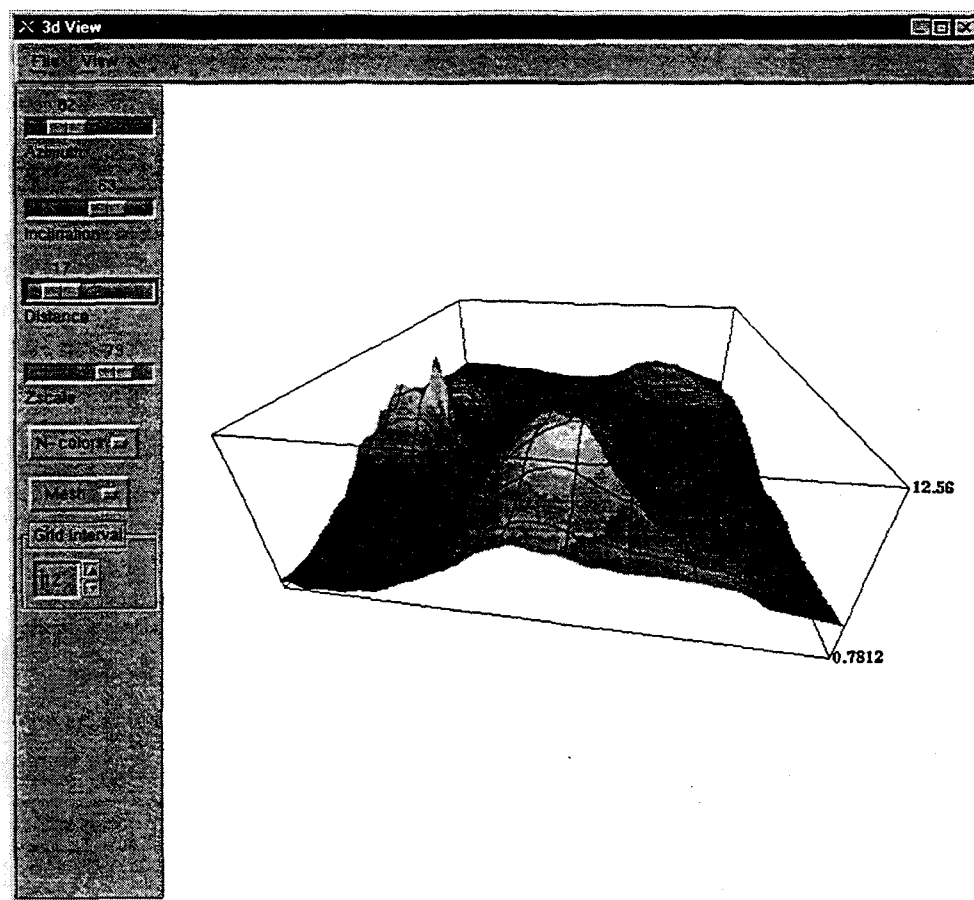


Figure 4 Topography displayed in 3 dimensions.

**SheEdit** also includes the facility for displaying the palette, which is posted when the “Scale...”-menu-item in the “View”-menu is activated. The following colour-legend appears:

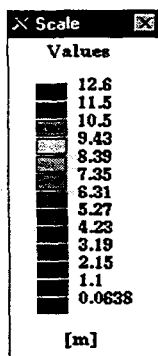


Figure 5 Colour legend with 13 colours.

Figure 5 displays the colour-legend for an autoscaled palette with 13 colours. If the user wishes to change the palette to a 10-colour palette with fixed values ranging from e.g. -8 to 10 it can be done using the "Palette Wizard". The "Palette Wizard" is accessed choosing the "Colours..."-menu-item in the "Colours"-menu and when the "Colours"-dialog is posted then by pressing "New...". The following dialog then appears:

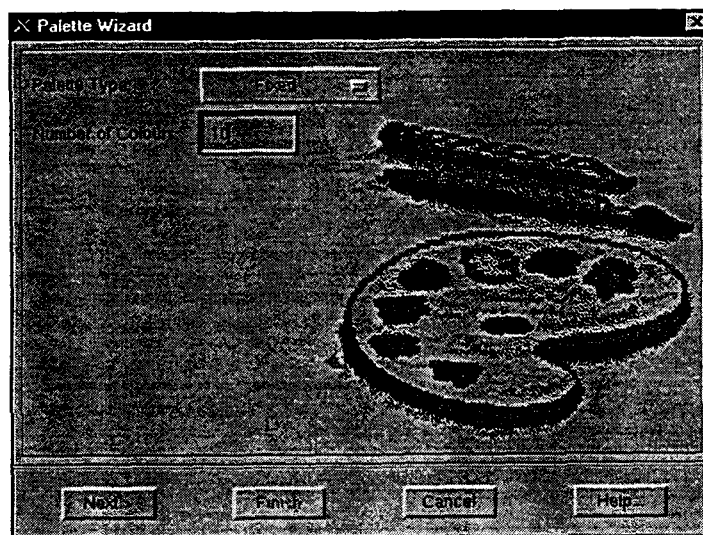


Figure 6 Palette wizard step 1.

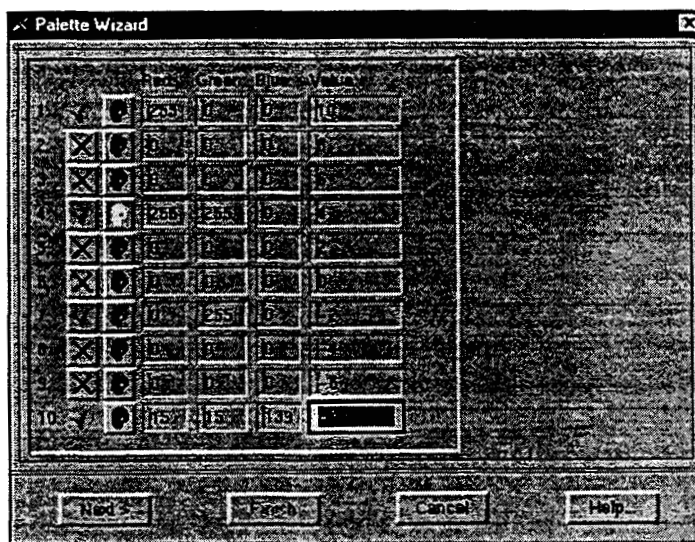


Figure 7 Palette wizard step 2.

Note that not all the colours are filled out. The colour-cells marked with a X are interpolated. As a minimum only the start and the end colour have to be specified. On the other hand all the corresponding values have to be filled out in descending order from the top. The result can be seen on Figure 8. The user accepts by pressing "Finish" or regrets by pressing "Cancel" which will result in the palette being discarded. The "Finish"-button becomes active when step 3, the preview mode" is reached.

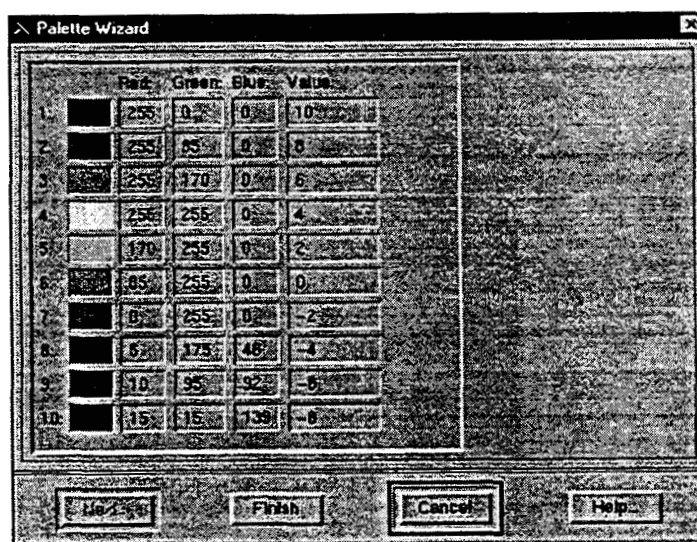


Figure 8 Palette wizard step 3.

Besides the "3x3 Mask" facility illustrated on Figure 3, which covers tabular editing of single values SheEdit, includes a wide variety of other editing facilities such as cluster editing, filtering and cropping. The cluster editing facility can be illustrated in the following way. The





“Select Polygon” function is used to define a polygon. The “Operations”-dialog is opened from the “Edit”-menu and assigning the selected values to 85 creates the following result.

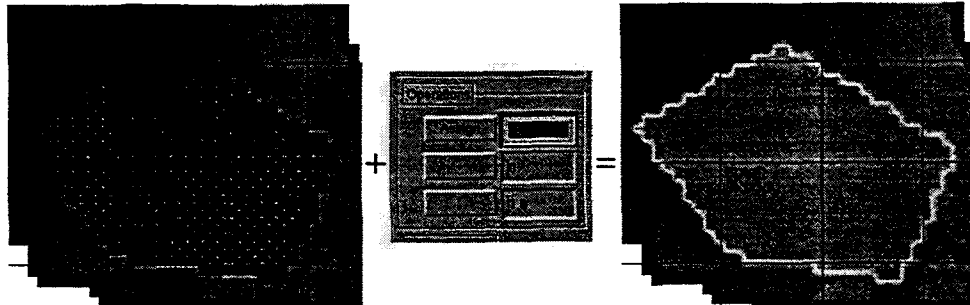


Figure 9 An example of cluster editing.

Filtering or cropping the data is done using the two dialogs shown on Figure 10. Filtering can be done on clustered data (Selected Area) or on the entire data set (Entire Area). The user can choose between 9 standard filters of which the three top ones smooths the data. Refer to the online documentation for a detailed description of the filters. Cropping the data is done using the “Cut Data”-dialog. Cropping the data means that the data outside the chosen area is discarded. This can be done by specifying the co-ordinates or cropping the data to the viewable area, meaning that everything not visible is discarded.

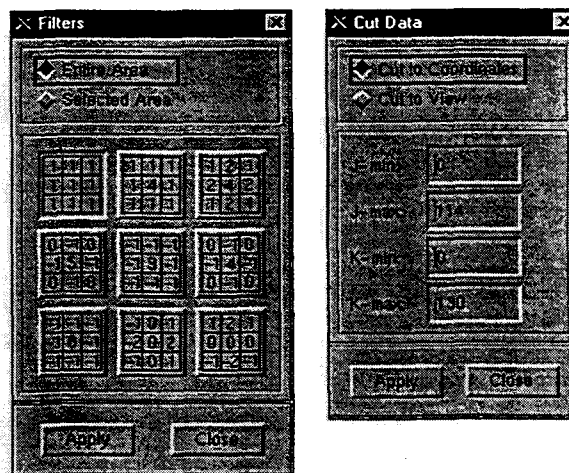


Figure 10 Dialogs for filtering and cropping the data.

The editing facilities are generally available from the “Edit”-menu.



# **MIKE SHE PP – User Manual**

## **Graphical Editing of River Network Data**





## CONTENTS

<b>1</b>	<b>GRAPHICAL EDITING OF RIVER NETWORK DATA .....</b>	<b>1</b>
1.1	Introduction.....	1
1.2	Pulldown Menus .....	3
1.2.1	The File Pulldown Menu .....	3
1.2.2	The Coords Pulldown Menu .....	4
1.2.3	The Colours Pulldown Menu .....	4
1.2.4	The View Pulldown Menu .....	6
1.2.5	The Network Pulldown Menu.....	7
1.2.6	The Edit Pulldown Menu.....	8
1.2.7	The Display Pulldown Menu .....	9
1.2.8	The Help Pulldown Menu .....	10
1.3	Left-hand Toolbar .....	10
1.4	Top Horizontal Toolbar .....	15
1.5	Popup Menu .....	15
1.6	Free Selection .....	17
1.7	General Editing Facilities.....	18
1.7.1	Editing of the longitudinal profile for a branch.....	19
1.7.2	Editing of cross-sections.....	21
1.7.3	Validation of the entire river network .....	23

122





# 1 GRAPHICAL EDITING OF RIVER NETWORK DATA

## 1.1 Introduction

The **RivEdit** is used to set-up the original MIKE SHE river model. MIKE SHE 1999 and newer versions use the MIKE 11 river modelling system. Regarding set-up of MIKE 11 reference is made to the "Short Introduction, Guide to getting started, Tutorial" to MIKE 11 1999. The **RivEdit** provides the user with a number of graphical viewing and editing facilities for MIKE SHE-files of type \*.dig and \*.rdf. Both file formats are ASCII-files and can be viewed/edited using a standard text editor though it is **NOT** recommended to do so, since the file isn't validated when saving the data. The dig-format contains raw digitised data and no relation between the points can be defined in this format. The rdf-format contains the digitised data but also allows complex relation between points to be defined like definition of river-branches. Moreover the rdf-format allows definition of cross-sections, external/internal boundaries and Q-stations.

When **RivEdit** is started the main graphical window appears and the "Open File Selection"-dialog is posted. When a file is selected, the data is then displayed in the main window as standard river network.

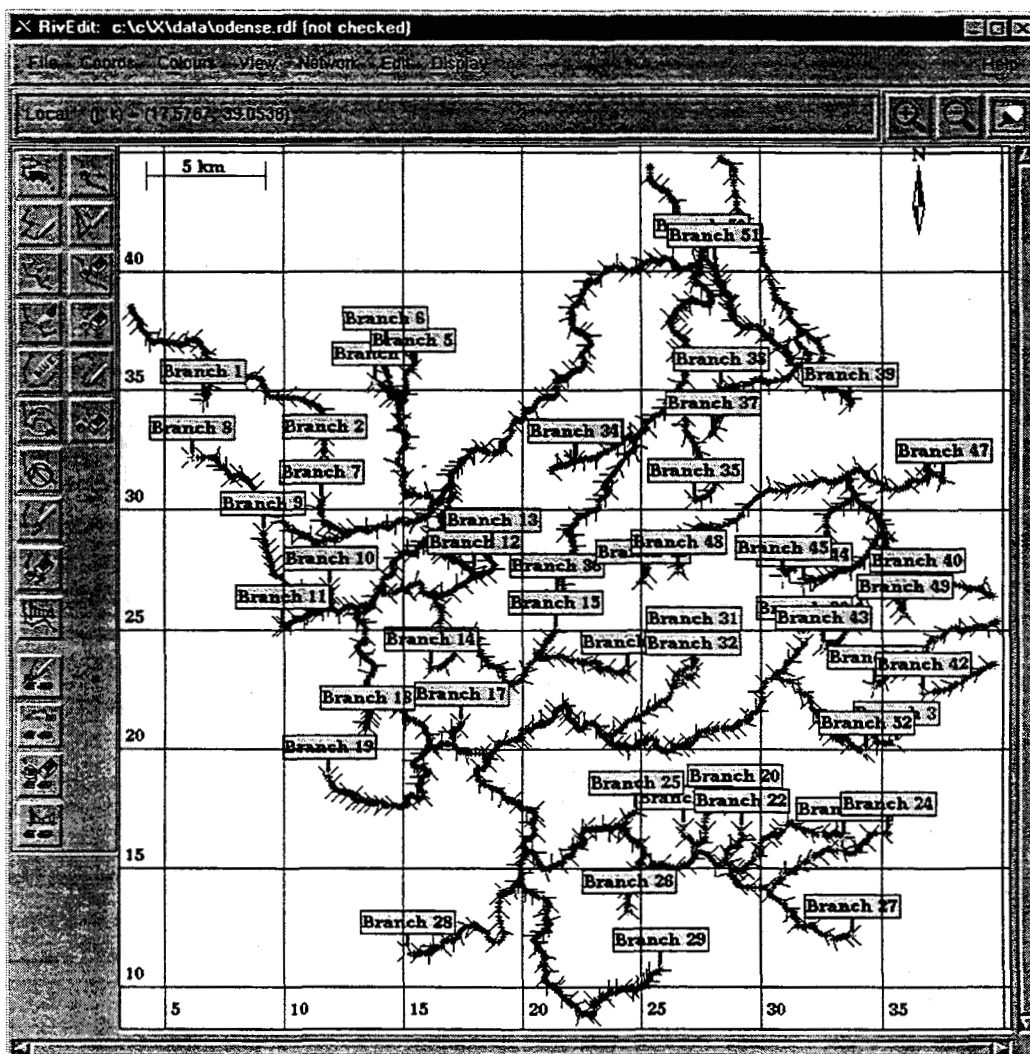


Figure 1 River network of Odense A displayed in RivEdit.

**RivEdit** features online help for all windows and dialogs, and this help is accessed by pressing the F1-key in the selected window or dialog. The general editing facilities can be accessed using the toolbar at the left-hand side of the main window or through the pulldown menus.



## 1.2 Pull-down Menus

### 1.2.1 The File Pull-down Menu

Activating the "File" button posts the following menu:

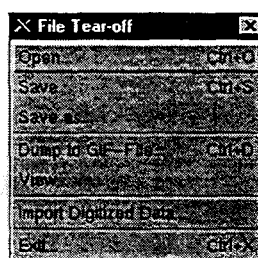


Figure 2 RivEdit's File-menu.

The entities are the following.

- Open... Activates the dialog for opening new files of various types. Available formats can be found by activating the "File Type" compobox in the "Open File Selection"-dialog. If this compobox is disabled the file format will be chosen automatically.
- Save Saves the data in the editor in the currently used data file. The user is not prompted for overwriting the file.
- Save as... Activates the dialog for saving the data in new files or in existing files. Available formats can be found by activating the "File Type" compobox in the "Save File Selection"-dialog. If this compobox is disabled the file format will be chosen automatically.
- Dump to GIF-File...  
Activates the dialog for specifying a name of the file in which the current view will be saved. When a valid file name is entered in the dialog and the "OK"-button is pressed, the current view is saved in GIF-file format.
- View... Activates the dialog for loading ASCII-text files for viewing purpose.
- Import Digitised Data...  
Activates the dialog for importing digitised data and adding it to the currently loaded data. If the digitised data is specified in a different geographical/UTM-co-ordinate system, the co-ordinates are automatically converted to





match the already loaded data set. If either the currently loaded data or the data in the dig-file in anonymous co-ordinates no conversion is done.

Exit... Activates the "Exit"-dialog.

### 1.2.2 The Coords Pull-down Menu

Activating the "Coords" button posts a menu with the following entities:

Local Selects equidistant grid in model co-ordinates. This option is default.

MIKE SHE

Selects equidistant grid in MIKE SHE model co-ordinates.

Geo Selects grid in geographical co-ordinates. This function is not available unless the data file contains valid information about geographical location and orientation.

UTM, SBF, DKS, OSGB, BTM, HKG, GAUSS-BOAGA, AMG, GAUSS-KRUGER, T.S.CASSINI

Selects grid in UTM (or in the chosen local UTM-like) co-ordinates. This function is not available unless the data file contains valid information about geographical location and orientation.

None Deselects grid.

Edit UTM Zone...

Activates the dialog for selecting UTM zone. The user can choose either auto-calculation of the UTM zone, which is based on the centre of the map, or a specific fixed UTM zone. The dialog is only available when UTM co-ordinates are selected.

Layout... Activates the dialog for selecting direction of the model co-ordinate axis. This function is never available in RivEdit.

### 1.2.3 The Colours Pull-down Menu

Activating the "Colours" button posts a menu with the following entities:



#### Red - Green - Blue

Selects a linear auto-scaled palette consisting of the colours red, green, blue and intermediate colours. The palette consists of 20 colours. This palette is default.

#### Black - Red - Yellow

Selects a linear auto-scaled palette consisting of the colours black, red, yellow and intermediate colours. The palette consists of 20 colours.

#### Wheat - Green - Blue

Selects a linear auto-scaled palette consisting of the colours wheat, green, blue and intermediate colours. The palette consists of 20 colours.

#### Lightblue - Darkblue

Selects a linear auto-scaled palette consisting of the colours from lightblue to darkblue. The palette consists of 20 colours.

#### White - Black

Selects a linear auto-scaled palette consisting of the colours from white to black. The palette consists of 20 colours.

#### White - Grey

Selects a linear auto-scaled palette consisting of the colours from white to grey. The palette consists of 20 colours.

#### Edit RGB...

Activates the dialog for colour editing. The user is prompted for an index in the colour table and the "Single RGB Select"-dialog is popped up. Note that if either the "Land Water 1"- or the "Land Water 2"-palette is selected the land colour cannot be edited in this dialog.

#### Grid Colour...

Activates the dialog for selecting the grid colour. The user can choose between 20 predefined non-editable colours.

Colours... Activates the advanced colour control dialog. The user can zoom in depth, choose between scaling types, edit and select delete values, load, save and create new palettes.



### 1.2.4 The View Pull-down Menu

Activating the "View" button posts a menu with the following entities:

**Coords** This button toggles monitoring of the co-ordinates for the mouse pointer on and off. When switched on the mouse pointer position is displayed in the text field right below the menu bar. The co-ordinates are displayed in the same co-ordinate system as selected in the "Coords"-menu. For dt2-, dt3- and T2-files the height z is displayed as well. The button is default switched on.

**Overview...**

Activates the window for displaying the overview (the entire data set) which includes options for changing the area by zooming or panning.

**Scale...**

Activates the window for displaying the colours and the related values. When double clicking on one of the colours the "Single RGB Select"-dialog for editing the RGB-values is popped up with the current colour. Note that the land, if present, cannot be edited.

**Zoom In**

Activates the zoom in function. The user then has to click, drag and release the mouse pointer to select the desired area as the new view.

**Zoom Out**

Activates the zoom out function, that zooms all the way out.

**Previous**

Recalls the previous zoom.

**Refresh**

Activates the function for redrawing the entire topography as well as overlays.

**Select Point**

Activates the select point function. The user moves the mouse pointer to the desired location and clicks in the map. This is used when the "3x3 Mask" function is active.

**Select Area**

Activates the select area function. The user then has to click, drag and release the mouse pointer to select the desired area. In this rectangular area all the grid points are selected. This function duplicates the "Select Rectangle" function.



#### Auto-scale...

Activates the dialog for scaling the area that contains the data. The area can be either expanded or shrunk in the four directions or scaled to fit the data.

### 1.2.5 The Network Pull-down Menu

Activating the "Network" button posts a menu with the following entities:

#### Edit Branch...

Activates the "Edit Branch"-window when a branch is selected. When the cursor is moved close to a branch it changes shape to a co-ordinate system. A new window for editing the profile is then opened by pressing the left-hand mouse button. Pressing 'ESC' exits the function without opening a new window. The function is identical with the "Edit Branch..." found in the first "Bitmap Column".

#### Name Branch...

Activates the "Name Branch"-dialog. The user can repeatedly specify a name and then apply it to a branch.

#### Auto-route Set-up...

Activates the "Auto-route Set-up-dialog. The different options to the auto-route algorithm are specified here.

#### Enter Level...

Activates the "Level in Point"-dialog. The user can repeatedly specify a level and then apply it to a point or retrieve the level from a chosen point.

#### Edit Cross Section...

Activates the "Edit Cross Section"-window when a cross section is selected. When the cursor is moved close to a point containing a cross-section it changes shape to a co-ordinate system. A new window for editing the cross section is then opened by pressing the left-hand mouse button. Pressing 'ESC' exits the function without opening a new window. The function is identical with the "Edit Cross Section..." found in the first "Bitmap Column".

#### Load Cross Section...

Activates the "Load Cross Section"-dialog. The dialog enables the user to load external ASCII-files (\*.crs) specifying a cross-section. The cross-section can then be applied graphically to a point with no cross section. This is



done after pressing the "OK"-button in the "Load Cross Section"-dialog. When the cursor is moved close to a point containing no cross section it changes shape to a floppy and a pair of reading glasses. The cross section is placed in the point by pressing the left-hand mouse button. Pressing 'ESC' exits the function without loading the data.

#### Save Cross Section...

Activates the "Save Cross Section"-dialog. The dialog enables the user to save external ASCII-files (\*.crs) specifying a cross-section. The cross-section is graphically retrieved from a point in the main map. This is done after pressing the "OK"-button in the "Save Cross Section"-dialog. When the cursor is moved close to a point containing a valid cross section it changes shape to a floppy and a pencil writing on it. The chosen cross section can then be written by pressing the left hand mouse button. Pressing 'ESC' exits the function without saving the data.

### 1.2.6 The Edit Pull-down Menu

Activating the "Edit" button posts a menu with the following entities:

#### External Boundary Cond. ...

Activates the "External Boundary Conditions"-dialog. The user can repeatedly specify external boundary conditions and then apply them to a branch, retrieve them from a branch or delete them.

#### Internal Boundary Cond. ...

Activates the "Internal Boundary Conditions"-dialog. The user can repeatedly specify external boundary conditions and then apply them to a point in a branch, retrieve them from a point in a branch or delete them.

#### Q-Stations...

Activates the "Q-Stations"-dialog. The user can repeatedly add or delete Q-stations in points that are members of a branch.

#### Branch Names...

Activates the "Branch Names"-dialog. The user can rename multiple branches in one action. The user can choose between all auto-generated names or all names.



### Topology Check...

Activates the "Topology Check"-dialog. The user can check the entire river set-up for errors and warnings in this dialog. The user can switch on/off the desired options that he/she wants to check for. In order to verify the entire river set-up all possible errors have to be switched on.

## 1.2.7 The Display Pull-down Menu

Activating the "Display" button posts a menu with the following entities:

**Points** This button toggles display of digitised points on and off. The button is default switched on.

**Branches** This button toggles display of branches on and off. The button is default switched on.

### Cross Sections

This button toggles display of cross sections on and off. The button is default switched on.

**Flow** This button toggles display of flow direction on and off. If no branches are displayed the display of flow direction is switched off independently of this button. The button is default switched on.

**Labels** This button toggles display of labels on branches on and off. The button is default switched on.

### External Boundaries

This button toggles display of external boundary conditions on and off. The button is default switched on.

### Internal Boundaries

This button toggles display of internal boundary conditions on and off. The button is default switched on.

### Q-Stations

This button toggles display of Q-stations on and off. The button is default switched on.

### Digitised Numbers

This button toggles display of digitised numbers for points on and off. The digitised numbers are not displayed if more than 50 points are shown inside the current view. The button is default switched on.



### 1.2.8 The Help Pull-down Menu

Activating the "**Help**" button posts a menu with the following entities:

**General...** Activates the "Help Text"-dialog. In general help can be obtained by pressing F1. The help system then searches recursively back until it finds an object with help attached. The help text is then displayed in the "Help Text"-dialog.

**About SheEdit...**

Activates the "About RivEdit"-dialog.

### 1.3 Left-hand Toolbar

The toolbar consists of a number of small icon buttons. Each of them activates functions that are used for selecting data. When the mouse cursor is moved inside a bitmap button a small flyby help displays the function of the button. This is used as the text in the following list of functions in the toolbar.



#### Auto-route Branch

This button activates the function for auto-routing branches. Only the starting and the end point are chosen. The branch is then automatically routed according to the options specified in the "Auto-route Set-up-dialog. Pressing 'ESC' exits the function without changing the data.



#### Define Branch

This button activates the function for manually routing each single point in a branch. One branch is routed by clicking the left-hand mouse button, dragging the cursor along the digitised points, and then releasing the button. Manually routing can be continued from the end of a previous branch. Pressing 'ESC' exits the function without changing the data.



#### Move Branch

This button activates the function for moving a branch. When the cursor is moved close to a branch it changes shape to a cross arrow. The branch can then be moved by pressing the left-hand mouse button, dragging the cursor to the desired location and then releasing the button. Pressing 'ESC' exits the function without changing the data.



#### Cut Branch into Two

This button activates the function for cutting a branch into two. When the knife cursor is moved close to a branch segment (line between two adjacent points) it changes shape to an active knife. The branch is then cut at the chosen location by pressing the left-hand mouse button. Pressing 'ESC' exits the function without changing the data.



#### Merge Branches

This button activates the function for merging two branches into one. When the glue cursor is moved close to the end of a branch (flow direction decides the start and the end point of a branch) glue comes out of the tube. The selected branch is then merged into another by clicking the left-hand mouse button, dragging the cursor to the start of the other branch and then releasing the mouse button. Pressing 'ESC' exits the function without changing the data.



#### Swap Flow Direction on Branch

This button activates the function for swapping flow direction on a branch. When the cursor is moved close to a branch it changes to two circular arrows. The flow direction can then be swapped by pressing the left-hand mouse button. Pressing 'ESC' exits the function without changing the data.



#### Delete Branch

This button activates the function for deleting a branch. When the cursor is moved close to a branch it changes shape to an eraser. The branch can then be deleted by pressing the left-hand mouse button. Pressing 'ESC' exits the function without changing the data.



#### Connect Branch to Another

This button activates the function for connecting a branch to another. When the cursor is moved close to either the end or the start point of a branch it changes shape to a pencil. The branch is then connected to another by clicking the left-hand

133





mouse button, dragging the cursor to a point in another branch and then releasing the mouse button. Note that a branch cannot be connected to either the start or the end point in another branch. Double clicking on the mouse button ends the function. Pressing 'ESC' exits the function without changing the data further.



#### Disconnect Branch from Another

This button activates the function for disconnecting a branch from another. When the cursor is moved close the start point of the connection it changes shape to an eraser. The connection is then deleted by pressing the left-hand mouse button. Double clicking on the mouse button ends the function. Pressing 'ESC' exits the function without changing the data further.



#### Edit Branch...

This button activates the function for graphically selecting a branch for editing of the longitudinal profile. When the cursor is moved close to a branch it changes shape to a co-ordinate system. A new window for editing the profile is then opened by pressing the left-hand mouse button. Pressing 'ESC' exits the function without opening a new window.



#### Add Cross Section

This button activates the function for adding a cross section in a point. When the pencil cursor is moved close to a point without a cross section it becomes active. A default cross section can then be added by pressing the left-hand mouse button. Cross sections can be added freely in points whether or not they form a branch. Double clicking on the mouse button ends the function. Pressing 'ESC' exits the function without changing the data further.



#### Move Cross Section

This button activates the function for moving a cross section. When the cursor is moved close to a point containing a cross section it changes shape to a cross arrow. The cross section can then be moved to another point by pressing the left-hand mouse button, dragging the cursor to



an empty point and releasing the mouse button. Double clicking on the mouse button ends the function. Pressing 'ESC' exits the function without changing the data further.



#### Delete Cross Section

This button activates the function for deleting a cross section. When the cursor is moved close to a point containing a cross section it changes shape to an eraser. The cross section is then deleted by pressing the left-hand mouse button. Cross sections can be deleted whether or not the points they are located in are members of a branch. Double clicking on the mouse button ends the function. Pressing 'ESC' exits the function without changing the data further.



#### Edit Cross Section...

This button activates the function for graphically selecting a cross section for editing. When the cursor is moved close to a point containing a cross section it changes shape to a co-ordinate system. A new window for editing the cross section is then opened by pressing the left-hand mouse button. Pressing 'ESC' exits the function without opening a new window.



#### Move Point

This button activates the function for moving a digitised point. When the cursor is moved close to a point it changes shape to a cross arrow. The point can then be moved by pressing the left-hand mouse button, dragging the cursor to the desired location and then releasing the mouse button. A point can be moved whether or not it is a member of a branch. Double clicking on the mouse button ends the function. Pressing 'ESC' exits the function without changing the data further.



#### Insert Point into Branch

This button activates the function for inserting a point into a branch. When the cursor is moved close to a point in a branch (except for the end point of the branch) it changes shape to a pencil. A free point can then be inserted after the chosen point. This is done by pressing the left-hand mouse button, dragging the cursor to the free point and then



releasing the mouse button. Double clicking on the mouse button ends the function. Pressing 'ESC' exits the function without changing the data further.



#### Exclude Point from Branch

This button activates the function for excluding a point from a branch. When the cursor is moved close to a point in a branch it changes shape to an eraser. The point can then be excluded from the branch by pressing the left-hand mouse button. Double clicking on the mouse button ends the function. Pressing 'ESC' exits the function without changing the data further.



#### Delete Level in Point

This button activates the function for deleting the level in a point. When the cursor is moved close a point containing a valid level it changes shape to an eraser. The level in the point can then be deleted by pressing the left-hand mouse button. When the level is deleted the point will contain a delete value. Double clicking on the mouse button ends the function. Pressing 'ESC' exits the function without changing the data further.



#### Add New Points

This button activates the function for adding new points. The cursor is changed to a pencil. Points containing delete values are added when the left-hand mouse button is pressed. Multiple points are added by dragging the cursor when the left-hand mouse button is pressed. Double clicking on the mouse button ends the function. Pressing 'ESC' exits the function without changing the data further.



#### Delete Free Points

This button activates the function for deleting free points. When the cursor is moved close to a free point (point not a member of a branch) it changes shape to an eraser. The point can then be deleted by pressing the left-hand mouse button. Multiple free points are deleted when the cursor is dragged over them when the left-hand mouse button is kept pressed. Double clicking on the mouse button ends the



function. Pressing 'ESC' exits the function without changing the data further.

## 1.4 Top Horizontal Toolbar

The top horizontal toolbar contains a message field for displaying various messages and monitoring of co-ordinates for the mouse cursor. Besides this message field it contains the following three buttons:



**Zoom In** This button activates the zoom in function. The user then has to click, drag and release the mouse pointer to select the desired area as the new view.



**Zoom Out** This button activates the zoom out function that zooms all the way out.



**Refresh** This button activates the function for redrawing the entire topography as well as overlays.

## 1.5 Popup Menu

Clicking with the right hand mouse button in the main map posts a popup menu containing the following entries:

**Zoom In** Activates the zoom in function. The user then has to click, drag and release the mouse pointer to select the desired area as the new view.

**Zoom Out**  
Activates the zoom out function, that zooms all the way out.

**Previous** Recalls the previous zoom.

**Refresh** Activates the function for redrawing the entire topography as well as overlays.

**Overview...**  
Activates the window for displaying the overview (the entire data set) which includes options for changing the area by zooming or panning. The "Overview"-window pops up with



the view centred around the cursor and is automatically destroyed when the cursor is moved outside the window.

#### Select Point

Activates the select point function. The user moves the mouse pointer to the desired location and clicks in the map. This is used when the "3x3 Mask" function is active.

#### Select Area

Activates the select area function. The user then has to click, drag and release the mouse pointer to select the desired area. In this rectangular area all the grid points are selected. This function duplicates the "Select Rectangle" function.



## 1.6 Free Selection

Double clicking with the left-hand mouse button in the main map activates editing of either cross sections or the longitudinal profile of a branch. When a double click is detected at some location in the main map, the program first searches for a cross section close to the cursor position. If a cross section is found the cross-section window is displayed. If no cross section is found close to the cursor position the program then searches for a branch. If a branch is found close to the cursor position the window for editing is displayed. Figure 3 shows an example where the user has double clicked on "Branch 32" and the longitudinal profile for this branch is then displayed in a separate window. When the cursor is moved inside the longitudinal profile window the branch is highlighted with a thick green line in the main map.

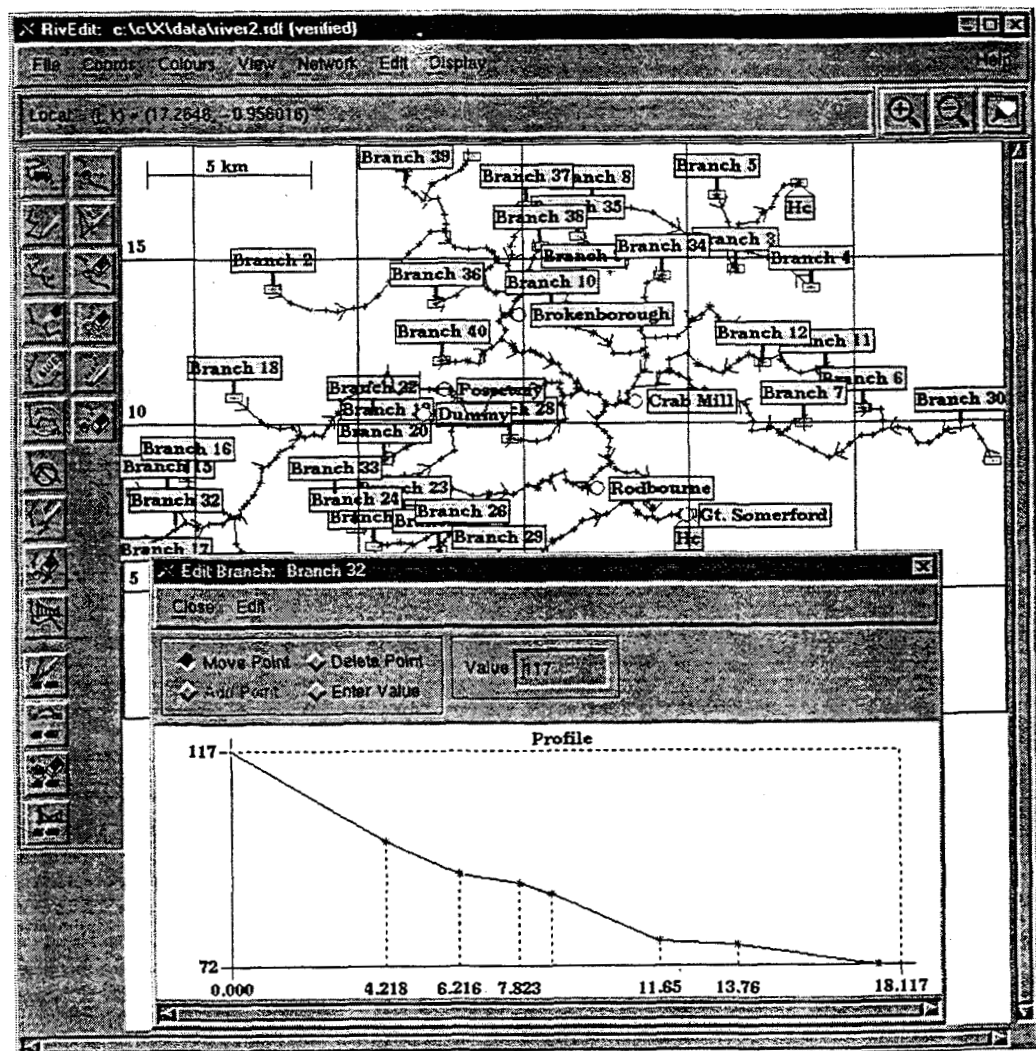


Figure 3 Longitudinal profile for a branch.



The "Edit Branch"-dialog displays the levels for the branch. The user can edit the longitudinal profile of the branch by changing the levels either by click and drag (Move Point-option) or by specifying the exact value (Enter Value). Illegal point can be excluded from the branch by using the "Delete Point"-option.

## 1.7 General Editing Facilities

RivEdit provides the user with a wide variety of graphical editing facilities in the main map. An example of this is the "Auto-route Branch" facility. The "Auto-route Branch" button is activated by the user, and the mouse cursor is then moved to the starting point. This point is selected by clicking without releasing and the cursor is then moved to the desired end point and released, and the branch is auto-routed as illustrated on Figure 4.

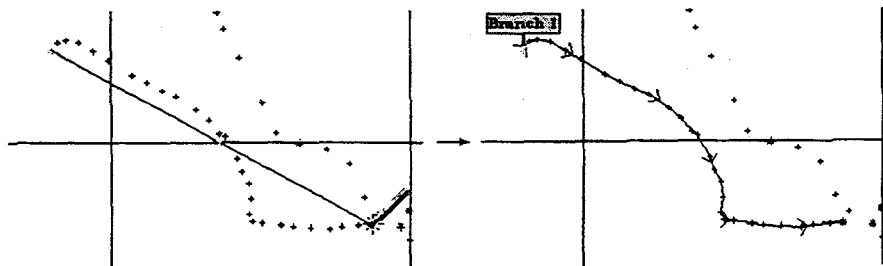


Figure 4 An example of how a river branch is auto-routed.

The other facilities that can be accessed from the toolbar works in a similar way. Specific differences can be found in the online help. RivEdit also includes the facility for displaying the palette, which is posted when the "Scale..."-menu-item in the "View"-menu is activated. The following colour-legend appears:

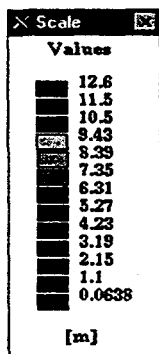


Figure 5 Colour legend with 13 colours.

Figure 5 displays the colour-legend for an auto-scaled palette with 13 colours. If the user wishes to change the palette to a palette with fixed values ranging from e.g. -8 to 10 it can be done using the "Palette



Wizard". The "Palette Wizard" is accessed choosing the "Colours..."-menu-item in the "Colours"-menu and when the "Colours"-dialog is posted then by pressing "New...".

Besides editing the data in the main window **RivEdit** includes editing in separate windows for the longitudinal profile of a branch and for the cross-sections

### 1.7.1 Editing of the longitudinal profile for a branch

The "Edit Branch"-window provides the user with facilities for editing longitudinal profiles of branches. The user can edit the nodes of the branch by moving them vertically, deleting them or specify certain values. Zooming can be performed if it is necessary.

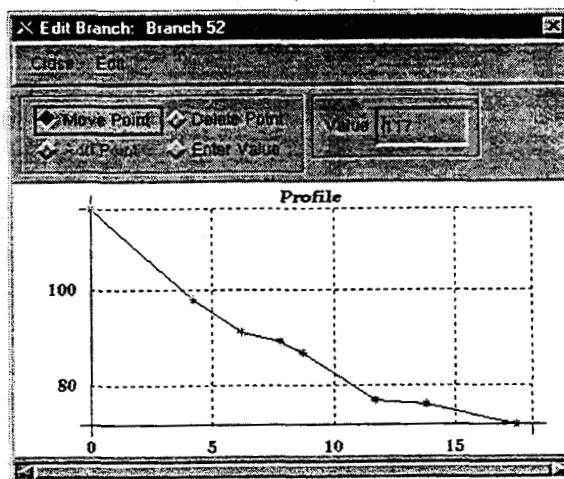


Figure 6 Longitudinal profile for a branch.

Figure 6 illustrates that there is direct access to four controls in the longitudinal profile window for a branch:

#### Move Point

This button controls the "Move Point" editing facility. When it is switched on the nodes in the branch can be moved vertically. This is done by moving the crosshair cursor close to the point. When the cursor changes shape to a double vertically arrow the point can be moved. This is done by clicking the left-hand mouse button and dragging the point to the right location. The current elevation can be monitored in the "Value"-edit control.

#### Delete Point

This button controls the "Delete Point" editing facility. When it is switched on the nodes can be deleted from the branch. This is done by moving the crosshair cursor close to





the point. When the cursor changes shape to an eraser the point can be deleted from the branch by clicking the left-hand mouse button.

#### Enter Value

This button controls the "Enter Value" editing facility. When it is switched on the nodes can be given a specific elevation. This is done by typing the elevation in the "Value"-edit control and then moving the crosshair cursor close to the point. When the cursor changes shape to a pencil and a number the point will be given the specified elevation by clicking the left-hand mouse button.

#### Value

This edit control allows the user to either specify or monitor the elevation of the nodes in the branch. When the "Enter Value" control is selected the edit control is used for specifying the elevation. When the "Move Point" or the "Delete Point" controls is selected the edit control displays the elevation of any point when the cursor is moved close to it.

Besides these direct accessible controls a number of items can be found in the pull-down menus:

#### Close

Closes the "Edit Branch"-window.

#### Edit

Activates the "Edit" menu with the following items.

#### Select New Branch

Activates the "Select New Branch"-function. This is done by moving the cursor close to the wanted branch in the main map and clicking the left-hand mouse button. The old branch displayed in the "Edit Branch"-window is then replaced by the new one.

#### Zoom In

Activates the "Zoom In"-function. The user clicks, drags and releases the left hand mouse button in the "Edit Branch"-window.

#### Zoom Out

Activates the "Zoom Out"-function.

#### Overlay Grid

This button toggles display of an overlay grid on and off. The button is default switched off.



## 1.7.2 Editing of cross-sections

The "Edit Cross"-window provides the user with facilities for editing cross-sections. The user can edit cross-section by moving existing points freely, by adding new points, by deleting points or by entering x- and z-co-ordinates for existing points. Zooming can be performed if it is necessary.

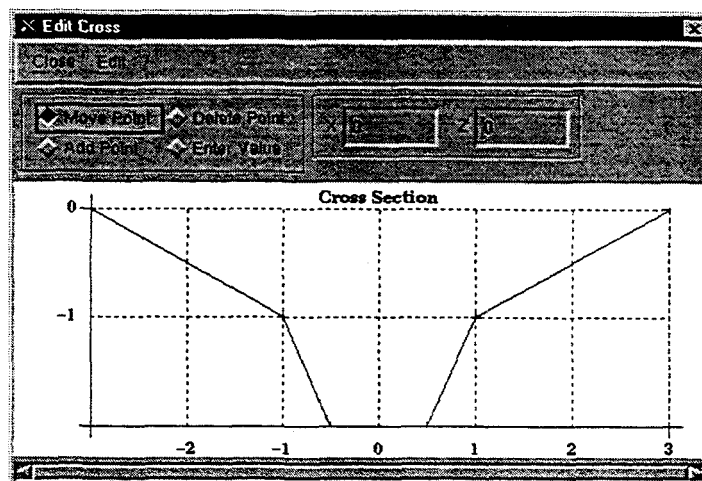


Figure 7 An example of a cross-section.

Figure 7 illustrates 6 direct accessible controls:

### Move Point

This button controls the "Move Point" editing facility. When it is switched on the points in the cross-section can be moved freely. This is done by moving the crosshair cursor close to a point. When the cursor changes shape to a 4-way arrow the point can be moved. This is done by clicking the left-hand mouse button and dragging the point to the right location. The current xz-position can be monitored in the "X"- and "Z"-edit controls.

### Add Point

This button controls the "Add Point" editing facility. When it is switched on new points can be added to the cross-section. This is done by clicking the left-hand mouse button at any location. The point is then inserted between the first adjacent set of points, where the first point has a smaller x-co-ordinate and the second one a larger x-co-ordinate.

### Delete Point

This button controls the "Delete Point" editing facility. When it is switched on points can be deleted from the cross-



section. This is done by moving the crosshair cursor close to the point. When the cursor changes shape to an eraser, the point can be deleted from the cross-section by clicking the left hand mouse button.

#### Enter Value

This button controls the "Enter Value" editing facility. When it is switched on points can be given a specific location. This is done by typing the xz-location in the corresponding "X"- and "Z"-edit controls and then moving the crosshair cursor close to the point. When the cursor changes shape to a pencil and a number the point will be given the specified xz-location by clicking the left hand mouse button.

X This edit control allows the user to either specify or monitor the x-co-ordinate of the points in the cross-section. When the "Enter Value"- control is selected the edit control is used for specifying the x-co-ordinate. When the "Move Point" or the "Delete Point" controls is selected the edit control displays the x-co-ordinate of any point when the cursor is moved close to it. The "X"-edit control is passive to the "Add Point"-control.

Z This edit control allows the user to either specify or monitor the z-co-ordinate of the points in the cross-section. When the "Enter Value"- control is selected the edit control is used for specifying the z-co-ordinate. When the "Move Point" or the "Delete Point" controls is selected the edit control displays the z-co-ordinate of any point when the cursor is moved close to it. The "Z"-edit control is passive to the "Add Point"-control.

Besides these direct accessible controls a number of items can be found in the pull-down menus:

Close Closes the "Edit Cross"-window.

Edit Activates the "Edit" menu with the following items.

#### Select New Cross

Activates the 'Select New Cross'-function. This is done by moving the cursor close to the wanted cross-section in the main map and clicking the left-hand mouse button. The old cross-section displayed in the "Edit Cross"-window is then replaced by the new one.



**Zoom In** Activates the "Zoom In"-function. The user clicks, drags and releases the left hand mouse button in the "Edit Branch"-window.

**Zoom Out**  
Activates the "Zoom Out"-function.

**Cross Characteristics...**  
Activates the function for opening "Cross Characteristics"-dialog.

**Save Cross...**  
Activates the function for saving the current cross-section in a separate file.

**Copy Manning = # to All**  
Activates the function for copying the Manning value of the current cross-section to all the other cross-sections.

**Copy Leakage = # to All**  
Activates the function for copying the Leakage value of the current cross-section to all the other cross-sections.

**Overlay Grid**  
This button toggles display of an overlay grid on and off. The button is default switched off.

### **1.7.3 Validation of the entire river network**

The validation of the river network is done through the "Topology Check"-dialog. The dialog provides the user with facilities for checking the entire set-up. The checks are divided up into two categories, namely errors and warnings. The user can choose to check for specific errors and warnings by toggling each of the check criterions on and off.

#### **Errors:**

- Increasing Branch Level
- Missing Boundary Conditions
- Missing Cross Sections
- Missing Levels in Free Endpoints
- Duplicate Names Used

145



#### Warnings:

- Unused Points
- Unused Cross Sections
- Locally Increasing Branch Level
- Decreasing x in Cross Sections

When the errors and warnings that the user wants to check for are selected the “Apply”-button is pressed. The entire river network is now verified. If any errors or warnings are found a list of these are generated. The errors will be displayed in the upper pane with red coloured text whereas the warnings will be displayed with yellow text in the lower pane. This is illustrated on Figure 8:

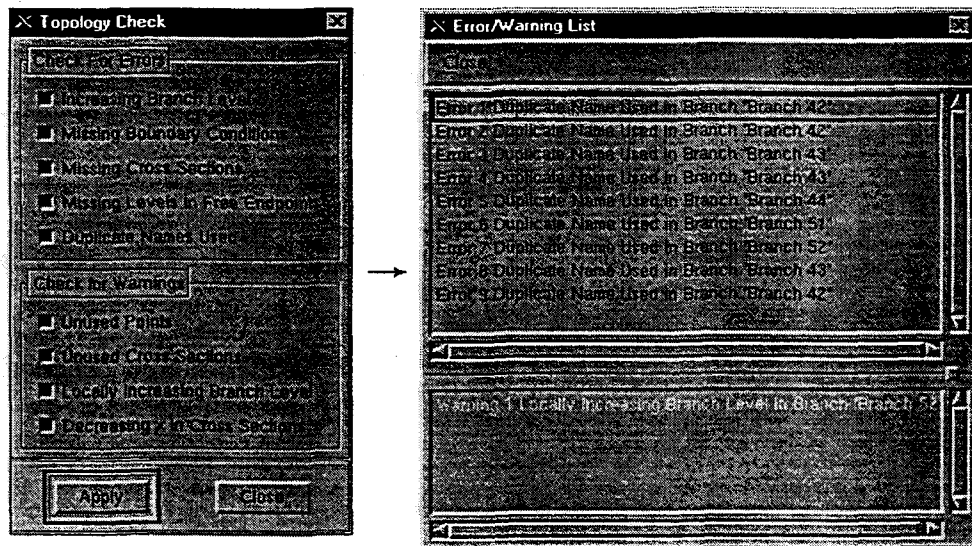


Figure 8 An example of a validation of a river network.

As seen in the example the verification generated 9 errors and 1 warning. In order to correct these errors and warnings the user can click or double click on one of the specified lines. One click highlights the erroneous object in the main map whereas a double click changes the view and jumps to the object. When all errors are removed from the set-up a verified-label is put on the data and stored in the file. However this label is instantly removed when further editing is done in the data. Thus the process of verification then has to be repeated.



# **MIKE SHE PP – User Manual**

## **Graphical Presentation**





## CONTENTS

<b>1</b>	<b>GRAPHICAL PRESENTATION.....</b>	<b>1</b>
1.1	General .....	1
1.2	Plotting of Time Series .....	5
1.3	Plotting of Matrix Data and Cross-sections .....	21
1.4	Animation of MIKE SHE Results .....	42







# **1 GRAPHICAL PRESENTATION**

## **1.1 General**

The MIKE SHE GRAPHICAL PRESENTATION (MSHE.GP) can be applied to produce coloured or rastered plots of input data or simulation results. Principally three different plot types can be plotted namely time series, aurally distributed data and cross-sections. Within this framework the MSHE.GP provides a highly flexible tool supporting a wide variety of different plot types and numerous combinations. Even though extensive error checking is carried out by MSHE.GP the high complexity of the system implies that meaningless plots can be generated. E.g. MSHE.GP does not prevent the user from producing a scatter diagram comparing the root depth for a certain crop with a river discharge at a certain gauging station. The following sections give a detailed description of the included plotting facilities.

### **General plotting specifications**

The MSHE.GP can be accessed either by selecting <GRAPHICS> on the main menu and clicking <GRAPHICAL PRESENTATION> on the succeeding menu or from the <Graphics> pull-down menu on some of the other MIKE SHE menu's.

The GRAPHICAL PRESENTATION main menu (menu G.1) includes data that must be specified for all plot types.

119



File Refresh Quit Help

menu G.1 GRAPHICAL PRESENTATION

Input file   file type

plotting parameters

device number

device format length  height

plots pr. page horizontal  vertical  plot scaling

start date year month day hour min dt video

end date

bottom text

legends on plot ☒ yes ☐ no

	no. selected	no. plots
<input type="button" value="1. select time series"/>	0	0
<input type="button" value="2. select grid values"/>	0	0

execute plotting program

plot status

Figure 1 The MIKE SHE GRAPHICAL PRESENTATION main menu.

### Input data file

The MSHE.GP allows you to plot data stored on a **flow input file** (fif) generated by the MIKE SHE SETUP PROGRAM, or on a **flow result file** (frf) from a water movement simulation. Simulation results can be plotted only from the frf whereas input data as hydrogeological parameters and surface topography, can be plotted both from frf and fif. The desired input data file is selected by choosing the input data file type and clicking the <select> button. When an input data file is selected it is read by MSHE.GP (loaded) and default values are displayed in menu G.1. Please note that whenever a new input data file is loaded, previous specifications that might be in the menus will be overwritten.



## **Output device**

The MSHE.GP supports numerous printers and plotters. The output devices available from your computer platform can be listed and selected by pressing the <select device> button. When a device has been selected the device format (length, height) is set automatically.

## **Plot dimensions- and location**

The MSHE.GP performs as default automatic scaling and location of the plots, but if the <plot scaling> button is switched to manual, user defined plot dimensions specified in menu G.1.1.a and G.1.2.a, will be applied. The manual option allows the user to define almost any desired plot layout. Generally it is most convenient to apply the automatic scaling option which scales the plots considering the following items:

- no. of plots per page,
- the output device format,
- inclusion/exclusion of legends, headers and bottom text and
- the catchment geometry (i.e. no of computational nodes in x and y direction).

## **Start and end date**

The start and end date is the default-plotting period (for time series plots) and if a frf-file has been loaded the plotting period will be initialised to the entire simulation period. For time series plots you can specify any desired plotting period for each plot on menu G.1.1.a. This period will always be valid for the actual plot overwriting the time specification on menu G.1.

## **Bottom text**

If a bottom text is specified it will be plotted at the bottom of each page together with the name of the input data file, the actual date and your userid on the computer. If the bottom text is empty none of the above mentioned will be plotted.



## Legends

The yes/no switch for legends is a global option (valid for all plots) that determines whether legends should be plotted or not, though legends will be plotted only if a legend text is specified for the actual data set, on menu G.1.1.a or G.1.2.a.

## Selection of time series or grid plots

The <select time series> and <select gridplots> is the entrance to menu G.1.1 and G.1.2, respectively where the data to be plotted are selected. How to select data will be described in details in Sections 1.2 and 1.3.

## Execution of MSHE.GP

When you have generated your plots the MSHE.GP is executed by clicking the <read and plot data> or the <plot data> pushbutton. <read and plot data> reads the input data file, plots the selected data and stores them on a temporary directory on the disk. If you may want to modify the plot slightly and plot the same data set again you can use the <plot data> option. The MSHE.GP will then read the specific data from the temporary directory, which will save some plotting time. Please note that the data in the temporary directory will be removed every time you quit the MIKE SHE system.

## Saving specifications

The <File> button in the menu bar allows you to save your plots in specification files, which can be loaded and reused. The MSHE.GD expects the specification files to be located on a subdirectory **PLOT** with the data file suffix **plt** (PLOT/\*.plt). It is recommended to observe this data file name convention.

## Refresh

The <Refresh> button unloads the input data file, clears all specifications and initialises to default values.



## 1.2 Plotting of Time Series

Plots of time series can be used for presentation and analysis of data varying in time. As no time varying data are stored in the **flow input file** time series can only be plotted from a **flow result file** or from a **type 0 data file** (see the **Data File Format** section in this manual).

### Selection of time series

If the button <select time series> on menu G.1 is pressed menu G.1.1 - SELECTION OF TIME SERIES appears (see Figure 2).

menu G.1.1 SELECTION OF TIME SERIES

File search path: TIMEP.Td

☐ display data types    ☐ display Q-stations  
☐ display line types

Select files

	plot number	data type	data file name	bx or rec. no.	iy	layer no.	item	line type	plot layout
1.									<input type="checkbox"/>
2.									<input type="checkbox"/>
3.									<input type="checkbox"/>
4.									<input type="checkbox"/>
5.									<input type="checkbox"/>
6.									<input type="checkbox"/>
7.									<input type="checkbox"/>
8.									<input type="checkbox"/>
9.									<input type="checkbox"/>
10.									<input type="checkbox"/>
11.									<input type="checkbox"/>
12.									<input type="checkbox"/>
13.									<input type="checkbox"/>
14.									<input type="checkbox"/>
15.									<input type="checkbox"/>

Figure 2 Menu G.1.1, Selection of time series.



Menu G.1.1 contains 15 **specification lines** each representing one time series, thus you can select a maximum of 15 time series that may be displayed in 15 different plots or e.g. in one plot with 15 time series. A time series is, in principle always selected by specifying the following items:

- the number of the plot that should contain the actual time series;
- a data type;
- the location in the grid network (ix,iy), or at a river branch (Q station) or as a data record in a type 0 data file (record number);
- a line type that determines how to plot the actual time series.

### Data types

The specifications needed to select a time series depends on the selected data type, e.g. a plot of the potential head (data type 15) requires specification of the location in the grid network (ix,iy,ilay) whereas plotting of a time series from a type 0 data file (data type 51) only needs specification of the number of the data record to be plotted, thus the system only prompts for the information needed for the actual data type. The available data types can be listed by clicking the <display data types> button on menu G.1.1.

## THE FOLLOWING DATA TYPES CAN BE PLOTTED:

### MIKE SHE RESULTS

1 - 34 : MIKE SHE WM simulation results.

35 - 47 : Display MIKE SHE WM simulation data types

- 41 : MIKE SHE AD simulation results: Concentration
- 42 : MIKE SHE PT simulation results: Number of particles
- 43 : MIKE SHE PT simulation results: Average age
- 44 : MIKE SHE PT simulation results: Transport time
- 45 : MIKE SHE PT simulation results: Number of registered particles
- 46 : MIKE SHE PT simulation results: Capture zones
- 47 : MIKE SHE AD simulation results: UZ Concentration

### MIKE SHE SETUP DATA

- 50 : Surface topography [m]
- 51 : Computational layers (elevations) [m]

### EXTERNAL FILES

- 51 : Plot a file. Values for a record within the specified period.

### ACCUMULATED VALUES

By adding 100-999 to the actual datatype you can plot accumulated values: duration curves or min/max values for a selected period.

These features are only relevant for datatypes 1-34 and 51.

The following types are available:

- +100 : Accumulated values for the entire plotting period.
- +200 : Yearly accumulation.
- +300 : Monthly accumulation.
- +400 : Change since start of plotting period.
- +500 : Duration curve.
- +600 : Yearly maximum.
- +700 : Monthly maximum.
- +800 : Yearly minimum.
- +900 : Monthly minimum.

## THE FOLLOWING DATATYPES ARE AVAILABLE:

Datatype	Explanation	Unit	Saved
1	precipitation	(mm/h)	X
2	actual evapotranspiration	(mm/h)	X
3	actual transpiration	(mm/h)	
4	evaporation from the soil surface	(mm/h)	
5	evaporation from intercepted storage	(mm/h)	
6	evaporation from ponded water	(mm/h)	
7	canopy storage	(mm)	X
8	infiltration to UZ	(mm/h)	
9	rate of change of storage in UZ	(mm/h)	
10	recharge to the saturated zone	(mm/h)	
11	evapotranspiration from the saturated zone	(mm/h)	
12	capillary calculated in the UZ - component	(mm)	X
13	accumulated error in the UZ - component	(mm)	
14	depth to phreatic surface	(m)	
15	head elevation in saturated zone	(m)	X
16	groundwater flow (8 items)	(?l/s/m)	X
17	depth of overland water	(m)	
18	overland flow in the x,y - direction	(m3/s)	
19	bypass flow in the unsaturated zone	(mm/h)	
20	depth of water in river	(m)	
21	snow storage	(mm)	X
22	river flow at specified stations	(m3/s)	
23	flow in the unsaturated zone	(mm/h)	
24	water content at each uz - node	(-)	
25	total inflow to river from overland	(m3/s)	
26	total inflow to river from aquifers	(m3/s)	
27	total inflow to river from drainage	(m3/s)	
28	effective water content in unsaturated zone	(-)	
29	total irrigation	(m3/s)	
30	irrigation intake from the river	(m3/s)	
31	irrigation pumping from wells	(m3/s)	

Figure 3 Available data types.



## Line types

A time series can be displayed in numerous different ways utilising different full or dashed lines, step diagrams, bar charts, scatter diagrams in almost any desired colour. The layout of each time series is selected by specifying an integer code representing a certain **line type**. The available line types can be listed by pressing the <display line types> button on menu G.1.1.

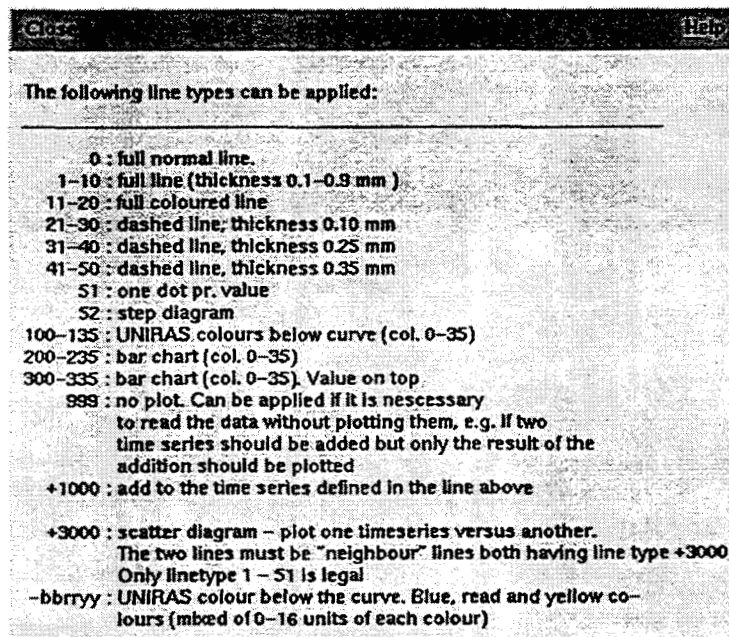


Figure 4 Available line types.

## Colours

The colours that can be applied are either UNIRAS default colours defined by an integer code (0-35) or a mixture of the three colours blue, red and yellow. The syntax for a mixture of these colours are:

- BBRRYY, where

BB is the amount of blue colour (0-16)

RR is the amount of red colour (0-16)

YY is the amount of yellow colour (0-16)

Please note the sign (minus) indicating that the colour is a mixture and not a default colour. The colour is selected by the line type (see Figure 4). Default colours and examples of mixed colours are shown in the **Uniras Colours** section in this manual.





In principle any data type can be combined with any desired plot type providing several thousand possible combinations which naturally cannot be treated individually. Figure 5 shows some examples of presenting time varying data.



Ok Clear Close Help

menu G.1.1 SELECTION OF TIME SERIES

☒ display data types    ☒ display Q-stations    Select files    File search path  
☒ display line types    TIMEP.To

	plot number	data type	data file name	bt or rec. no.	ly	layer no.	item	line type	plot layout
1.	1	1		5	5			230	✓ +
2.	2	2		5	5			52	✓ +
3.	3	15		8	5	1		21	✓ +
4.	3	15		5	4	1		24	✓
5.	3	15		7	7	1		27	✓
6.	4	22		1				-010101	◆ +

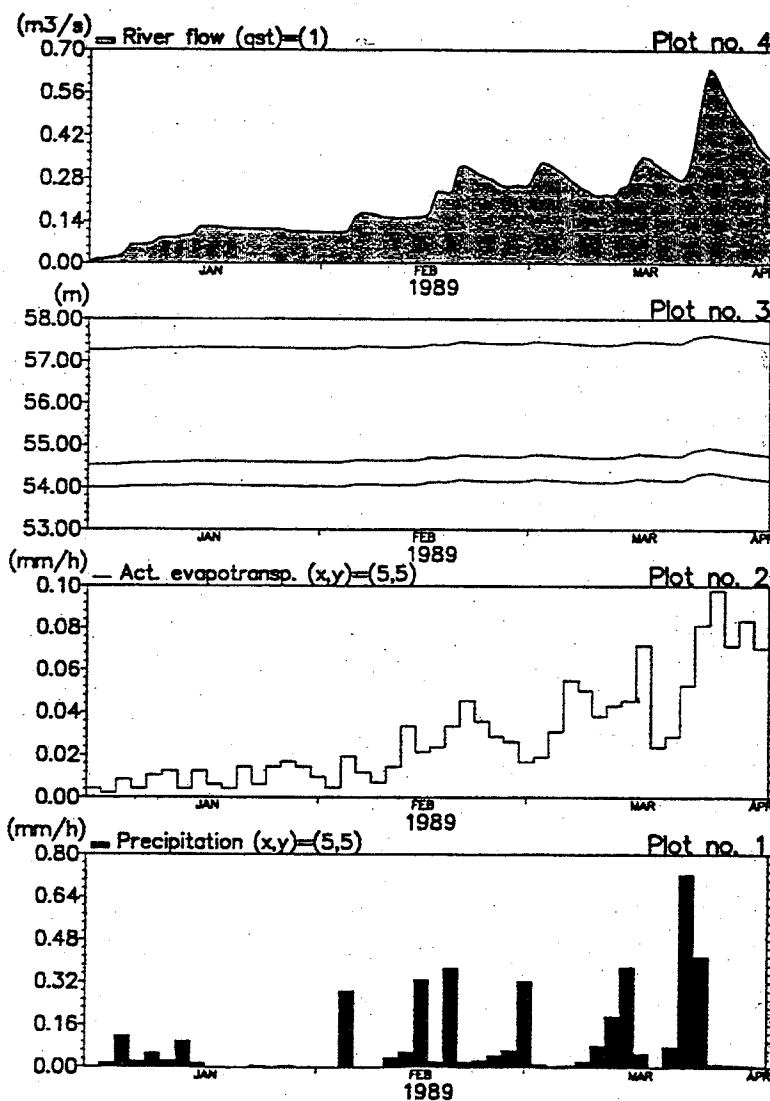
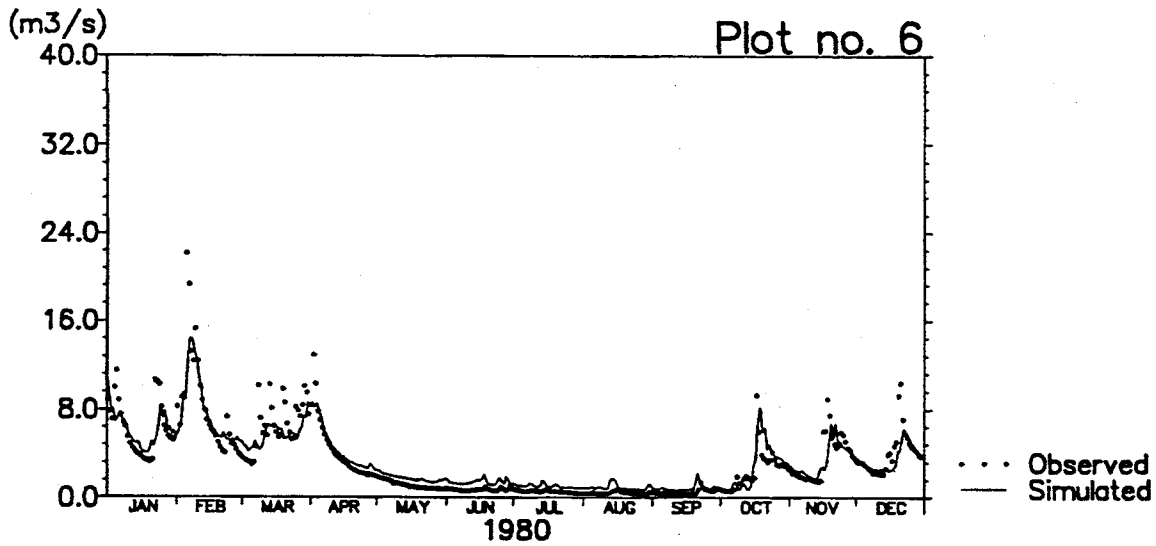
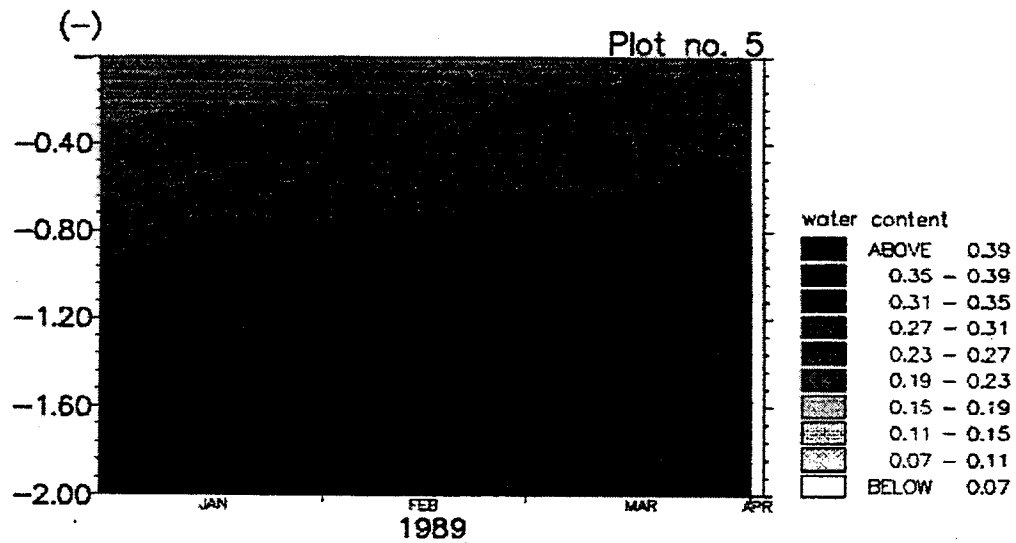


Figure 5 Examples of time series plots of different data types using different line types.



	plot number	data type	data file name	lx or rec. no.	ly	layer no.	Item	line type	plot layout
1.	1	22		1				1	◆ +
2.	1	51	TIME/qod.T0	1				51	▼



	plot number	data type	data file name	lx or rec. no.	ly	layer no.	Item	line type	plot layout
1.	1	24		5	5			0	▼ +

Figure 5 Examples of time series plots of different data types using different line types (cont.).



- Plot 1 : Precipitation rate (data type 1) plotted as a bar chart using UNIRAS default colour number 30 (line type 230).
- Plot 2 : Actual Evapotranspiration rate (data type 2) plotted as a step diagram (line type 52).
- Plot 3 : Potential head (data type 15) at three different locations in the grid network using different dashed lines (line type 21, 24 and 27).
- Plot 4 : River flow (data type 22) at river station 1 using a UNIRAS colour mixture below the curve, (line type -010101).
- Plot 5 : Moisture content in an unsaturated soil profile (data type 24). For data type 24 a colour scale can be specified on menu G.1.1.a. This is done as for grid plots explained in Section 1.3.
- Plot 6 : Comparison between simulated (data type 22) and observed river flow (data type 51) plotted as line type 1 and 51 respectively.

### **Modifying the plot layout**

The specification lines on menu G.1.1 only defines the time series to be plotted and also how to plot it (data type and line type). If you do not specify anything else the MSHE.GD will use defaults for plot geometry, text legends, headers etc. If you need to change those defaults you must click the button at the right end of the actual specification line to enter menu G.1.1.a.

159



Apply Close Help

menu G.1.1.a PLOT LAYOUT for plot 1 in line 1

X axis: year month day hour min  
start date 1981 1 1 0 0  
end date 1981 2 1 0 0

Y axis: Ymin Ymax Delta dec.  
0 0 0 2

Y axis text (m)

data - mult. factor add. constant  
transformation 1 0

plot header

legend text Head elevation (x,y,lly)-(12,94,1)

plot geometry

Figure 6 Menu G.1.1.a, plot layout.

Please note that the specifications on menu G.1.1.a will not be valid until the <Apply> button is pressed. The <Close> button will close the menu ignoring any modifications.

### X axis

The plotting period for the actual plot can be modified by changing the start and end date.

### Y axis

The Y-axis is defined by a minimum and maximum value (Ymin, Ymax), the step between each division (Delta) and the number of decimals on the axis values (dec). If Ymin and Ymax is specified to zero automatic scaling of the Y-axis is performed, depending on the magnitude of data set to be plotted. If Delta is zero the Y-axis will have five divisions each marked with a data value. The plots in Figure 5 are all produced using automatic axis scaling and Delta equal to zero.



## Data transformation

A time series can be transformed by multiplying or adding the data set with a constant value. This is a convenient option e.g. if you want to transfer a data set into another unit.

## Headers and legends

A default legend is generated automatically depending on the data type and location, whereas the header is left empty. You can of course specify headers and legends as desired.

## Plot geometry

If you want to modify the default plot size and location you must click the <plot geometry> button on menu G.1.1.a. Please observe to switch plot scaling to <manual> on menu G.1 before changing the plot geometry. Otherwise your specifications will be overwritten when plotting the data or if the <Ok> button on menu G.1.1 is pressed.

## Text heights

The size of the letters in headers, legends etc. can be changed by specifying a desired text height.

## Grid layout

You can modify the "basic layout" of the plot by switching different plot elements on or off. Figure 7 defines the terms that are used for the single elements forming a plot.

	width	height	Xo	Yo
Size and location	50	59.33	25	15

	x-text	y-units	y-ads	headers	legends
Text heights	2.5	2.5	2.5	5	2.5

	frame	grid	ads	x-text	y-text	sub markers
Grid layout	Off	Off	On	On	On	On

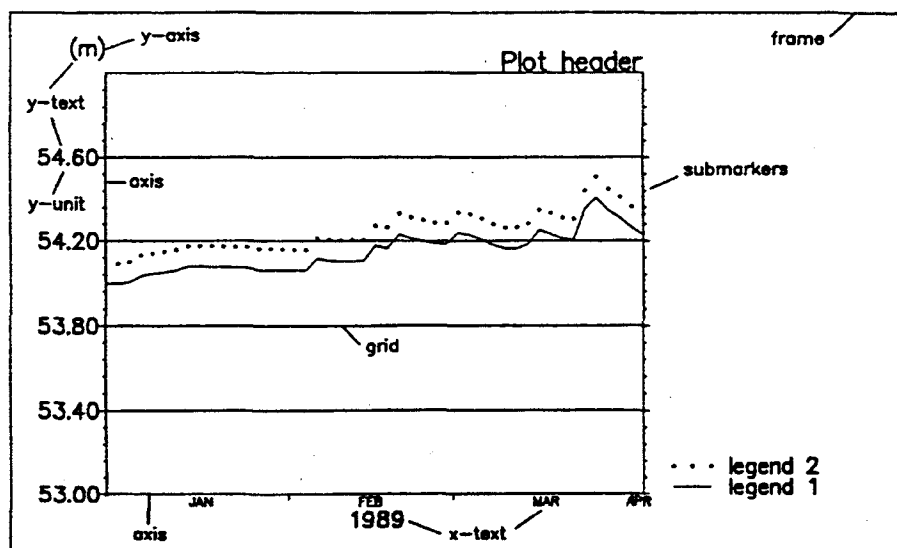


Figure 7 Plot geometry and explanation of the applied terms.

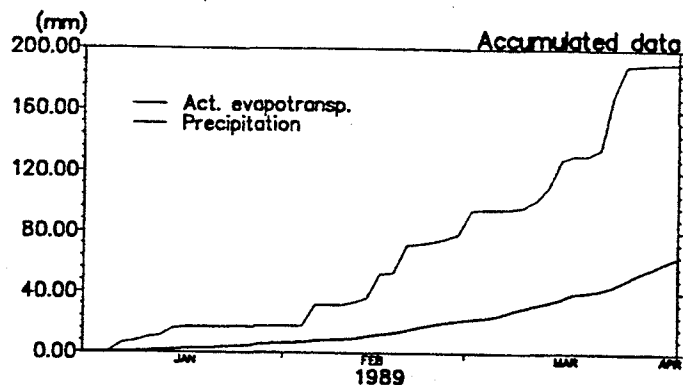
If you have defined more than one time series in a plot certain specifications as axis, plot dimensions and headers are unique for the single plot, hence they cannot be specified for each time series in the same plot. MSHE.GP will therefore only allow those specifications to be entered once, namely in the first specification line that enters the plot layout menu, menu G.1.1.a. The line that holds the plot specific definitions are marked with a '+' on menu G.1.1. Legend text and data transformation parameters can be specified for each single time series in the plot.

### Special plotting facilities

The plots in Figure 5 shows how to produce relatively simple plots of time varying data. The MSHE.GP also includes useful facilities to process and analyse a data set as:

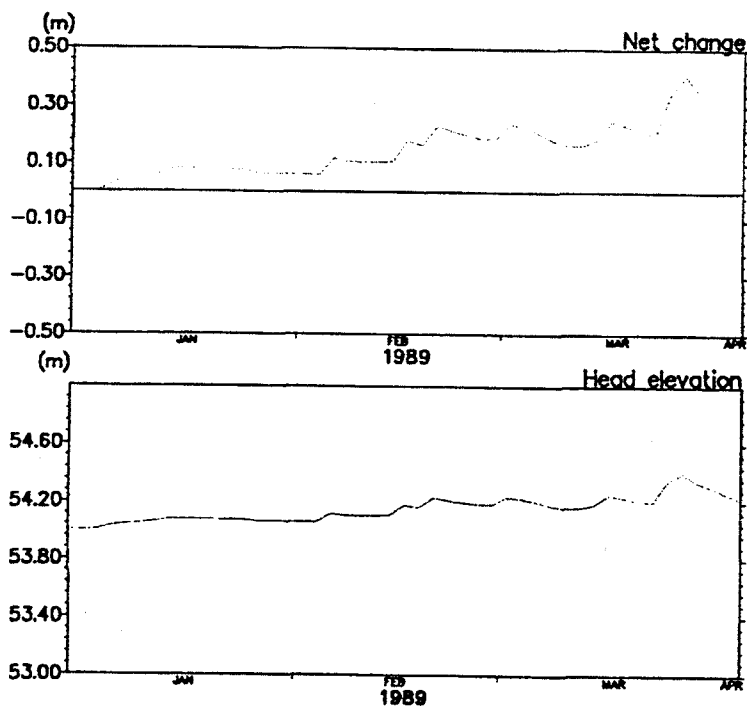
- accumulating data over a certain period of time;
- finding and plotting minimum and maximum values;
- producing duration curves;
- comparing two time series in a scatter diagram.
- adding or subtracting two time series.

Such plots are produced by adding a multiplier of hundred to the data type that should be plotted (see Figure 3), and as for ordinary time series plots data type and line type can be combined as desired. The plots in Figure 8 - Figure 11 show some of the above mentioned facilities.



	plot number	data type	data file name	lx or rec. no.	ly	layer no.	Item	line type	plot layout	
1.	1	101		5	5			1	✓	+
2.	1	102		5	5			2	✓	

Figure 8 Accumulated evapotranspiration and precipitation.

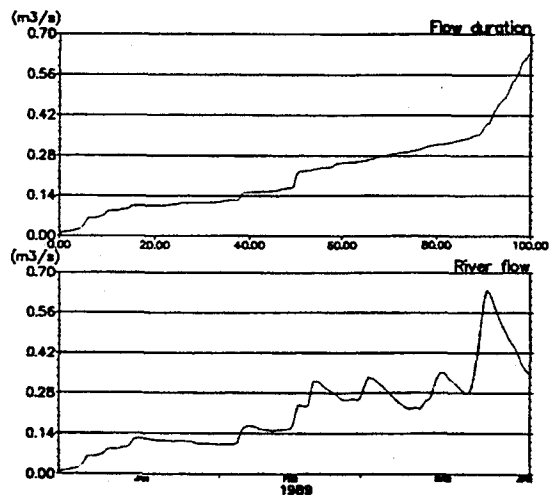


	plot number	data type	data file name	lx or rec. no.	ly	layer no.	Item	line type	plot layout	
1.	1	15		5	5	1		22	✓	+
2.	2	415		5	5	1		23	◆	+

Figure 9 Head elevation and net change since start of the plotting period.

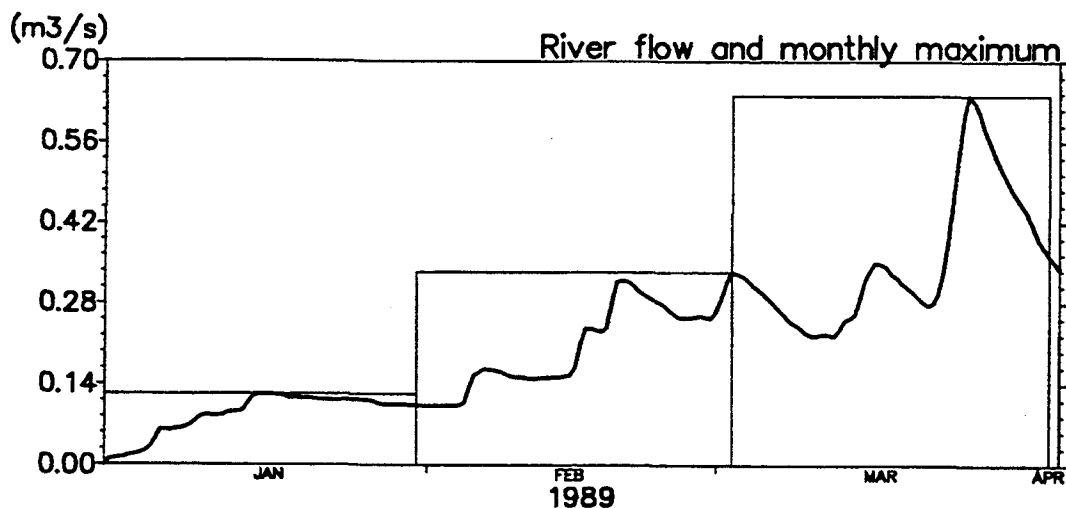
163





	plot number	data type	data file name	lx or rec. no.	ly	layer no.	Item	line type	plot layout	
1.	1	22		1				25	◆	+
2.	2	322		1				25	▼	+

Figure 10 River flow and flow duration curve.



	plot number	data type	data file name	lx or rec. no.	ly	layer no.	Item	line type	plot layout	
1.	1	722		1				200	▼	+
2.	1	22		1				3	◆	

Figure 11 River flow and monthly maximum.

### Adding subtracting and "integrating" time series

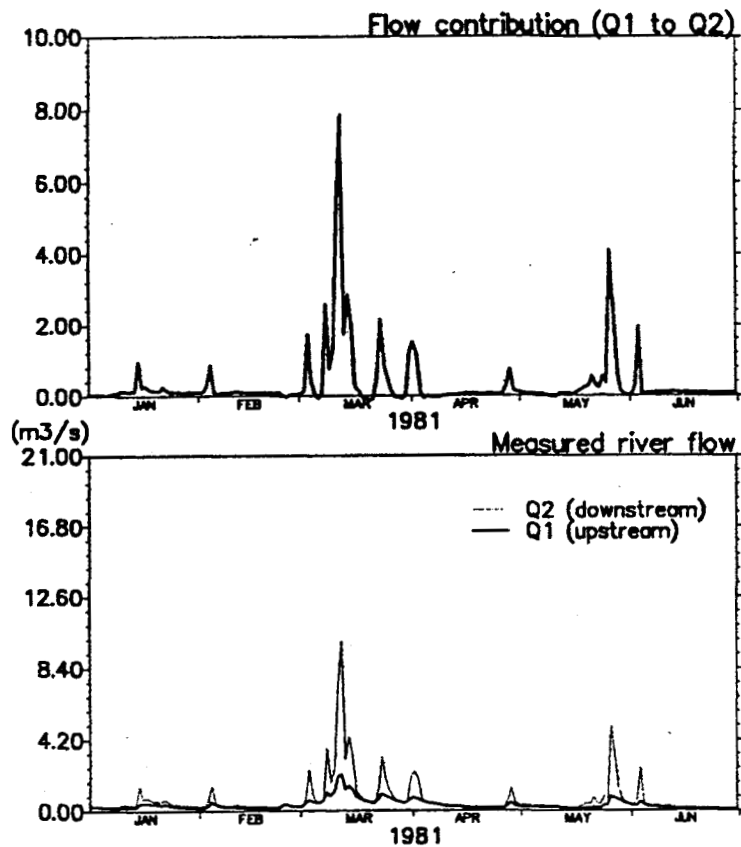
In Figure 12 is given an example that utilises the facility for adding two time series.



The example in Figure 12 include two river gauging stations, Q1 and Q2, located on the same branch, where Q1 is at a location upstream Q2. The first plot shows the river flow at the two stations, plotted from the type 0 data file, TIME/riverobs.T0. Q1 is data record no. 2 and Q2 is record no. 3 in the data file. Plot no. 1 is defined in specification lines 1 and 2. The second plot shows the contribution to the river flow from Q1 to Q2. The plot requires two specification lines. Line 3 includes the data from the downstream station Q2, plotted with line type 999 implying that the data should be read but not plotted. Line 4 includes the data from Q1 plotted as line type 1003 (3+1000) which means that the data should be added to the data specified in the specification line above. Using a multiplication factor equal to -1.0 implies that the time series specified in line no. 4 will be subtracted from the one in line no. 3 leading to the result  $Q2 - Q1$ .

Integration of for instance precipitation inside a catchment area is also possible. This is done by entering plot number, and plot type as usual and then specify a zero for 'ix'. The edit field "file name" is then opened and here you should enter the name of a grid code data file which has a code value of 1 in the grids you want to integrate over and 0 outside. This is a useful tool when examining water balances in a catchment.

165

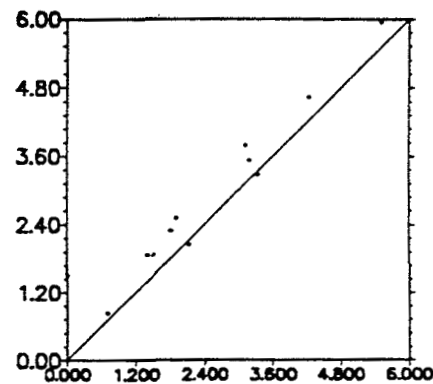


plot number	data type	data file name	lx or rec. no.	ly	layer no.	Item	line type	plot layout	
1.	<input checked="" type="checkbox"/> S1	TIME/r/verobs.T0	2				2	✓	+
2.	<input type="checkbox"/> S1	TIME/r/verobs.T0	3				22	✓	
3.	<input type="checkbox"/> S1	TIME/r/verobs.T0	3				888	✓	
4.	<input type="checkbox"/> S1	TIME/r/verobs.T0	2				1003	✓	+

Figure 12 Adding time series and plotting the result.

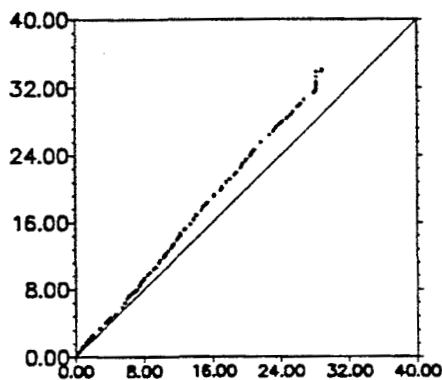
### Scatter diagrams

A scatter diagram can be applied to plot one time series versus another. You could e.g. plot observed versus simulated river discharge, or as shown in Figure 13 the monthly accumulated precipitation from two gauging stations. In order to plot two time series in a scatter diagram, 3000 must be added to the line type for both time series, and the two time series should be specified in "neighbour lines".



	plot number	data type	data file name	lt or rec. no.	ly	layer no.	item	line type	plot layout	
1.	1	351	TIME/prdreal.T0	1				3051	✓	+
2.	1	351	TIME/prdreal.T0	2				3051	✓	

Figure 13 Scatter diagram of monthly accumulated precipitation for two precipitation gauging stations.



	plot number	data type	data file name	lt or rec. no.	ly	layer no.	item	line type	plot layout	
1.	1	151	TIME/prdreal.T0	1				3051	✓	+
2.	1	151	TIME/prdreal.T0	2				3051	✓	

Figure 14 Double mass curve for two precipitation gauging stations.

In Figure 14 is shown the accumulated precipitation over a certain period of time, for the same two stations as in Figure 13 (double mass curve).

### AD- and PT-results

Plotting time series of transport simulations results (AD or PT simulation results) are a little more complex than the relatively simple plots in Figure 5. In menu G.1 an input file or a result file must be



selected even though the intention is to plot AD- or PT-data. The reason is that the grid description etc. is taken from this file, therefore the file you select must correspond to the transport simulation file you want to look at.

In addition to choosing one of the data types for transport simulations results in menu G.1.1 you must select the transport simulation result file (.trf) to be used. These .trf files are placed in '%sheres%\ad' directory, which on a standard PC installation would be 'C:\she524\res\ad', and the whole path including the file name is necessary for the MSHE.GD programme.

Specification of the location in the grid network is required and plotting a time series of the concentrations from an AD-simulation (data type 41 or 47) the desired species number must be specified too.

The menu system is not capable of automatically finding the start and end date of the transport simulation, therefore the plotting period must be specified in the plot layout menu. The dates you will find are the start and end date of the water movement simulation, taken from the input file in menu G.1. The period you specify must be within the start and end date of the transport simulation, or an error message will occur, but not until MSHE.GD is executed.

In Figure 15 a plot showing the number of particles as function of time together with the menus needing extra attention when specifying a time series plot of transport simulation results.

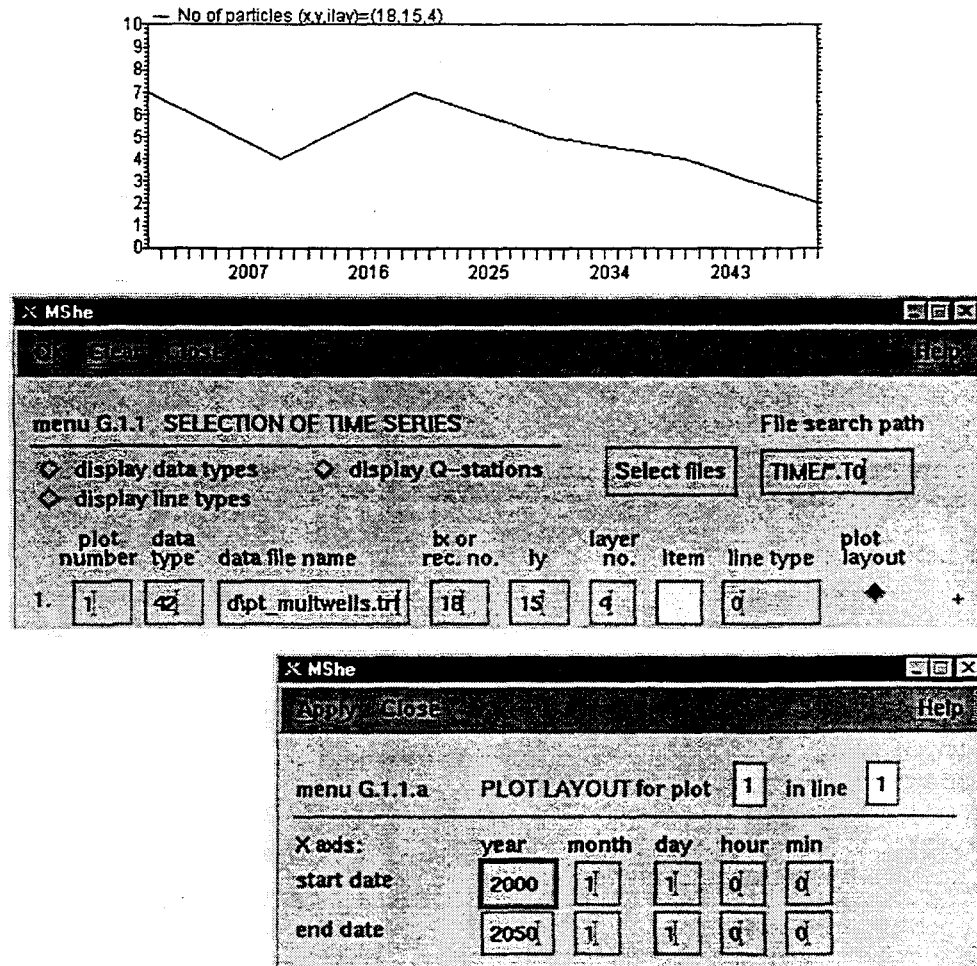


Figure 15 Time series of transport simulation results and menu parts to pay attention.

### 1.3 Plotting of Matrix Data and Cross-sections

The procedure for selecting and plotting spatially distributed data is very similar to the procedure for time series, and the hierarchy and layout of the menu are almost alike. As for time series the input can be read from flow result file and flow input file, though plotting from a flow input file is much more useful when plotting spatially distributed data, e.g. in connection with checking the representation of a geological model into the grid network.



## Selection of data

If the button <select grid values> on menu G.1 is pressed menu G.1.2 - SELECTION OF GRID VALUES appears.

	plot number	data type	filename	layer no.	item	plot type	plot layout
1.							◆
2.							◆
3.							◆
4.							◆
5.							◆
6.							◆
7.							◆
8.							◆
9.							◆
10.							◆
11.							◆
12.							◆
13.							◆
14.							◆
15.							◆

Figure 16 Menu G.1.2, Selection of grid values.

As it appears from Figure 16 menu G.1.2 also contains 15 specification lines and similar to the time series plots the essential specifications are a data type and a plot type.

## Data types

The data types available for grid plots can be listed by pressing the <display data types> button on menu G.1.2.



Cross

Help

THE FOLLOWING DATA TYPES CAN BE PLOTTED:

MIKE SHE RESULTS

1-34 : MIKE SHE WM simulation results

Display MIKE SHE WM simulation data types

41 : MIKE SHE AD simulation results. Concentration

42 : MIKE SHE PT simulation results. Number of particles

43 : MIKE SHE PT simulation results. Average age

44 : MIKE SHE PT simulation results. Transport time

45 : MIKE SHE PT simulation results. Number of registered particles

46 : MIKE SHE PT simulation results. Capture zones

MIKE SHE SETUP DATA

60 : Surface topography [m]

61 : Computational layers (Elevations) [m]

62 : Horizontal hydraulic conductivity [m/s]

63 : Vertical hydraulic conductivity [m/s]

64 : Unconfined storage coefficient [-]

65 : Confined storage coefficient [-]

66 : Drainage level [m]

67 : Drainage time constant [1/s]

68 : Manning number on ground surface [m<sup>1/3</sup> / sec]

69 : Detention storage [mm]

70 : Potential evapotranspiration stations (code values)

71 : Precipitation stations (code values)

72 : Vegetation types (code values)

73 : UZ Soil profiles (code values)

74 : UZ classification codes (code values)

75 : Temperature stations (code values)

76 : Paved areas (code values)

77 : Bypass codes (code values)

78 : Boundary grid codes (code values)

79 : Flow boundaries (Item 1-C,2-Y) [m/s]

80 : Gradient boundaries (Item 1-C,2-Y) [-]

EXTERNAL FILES

51 : Plot a 10 file. Location and value for a record at a specified output data

57 : Plot a 12 file

59 : Plot a dig file. Point/line polygon in colour depending either on selection or on 1- or code-value

61 : Plot a dig file. Point/line polygon in specified colour and 1- or code-value written at each figure  
(note! dig files type 23 is always plotted as river points)

62 : Plot a text from dig file. File formatted (code x y z text) list is plotted at (x,y) for code equal 1 or 3

Cross

Help

THE FOLLOWING DATATYPES ARE AVAILABLE:

Datatype	Explanation	Unit	Saved
1	precipitation	(mm/h)	X
2	actual evapotranspiration	(mm/h)	X
3	actual transpiration	(mm/h)	
4	evaporation from the soil surface	(mm/h)	
5	evaporation from intercepted storage	(mm/h)	
6	evaporation from ponded water	(mm/h)	
7	canopy storage	(mm)	X
8	infiltration to UZ	(mm/h)	
9	rate of change of storage in UZ	(mm/h)	
10	recharge to the saturated zone	(mm/h)	
11	evapotranspiration from the saturated zone	(mm/h)	
12	epsilon calculated in the UZ-component	(mm)	X
13	accumulated error in the UZ-component	(mm)	
14	depth to phreatic surface	(m)	
15	head elevation in saturated zone	(m)	X
16	groundwater flow (8 hours)	(l/min)	X
17	depth of overland water	(m)	
18	overland flow in the x,y-direction	(m <sup>3</sup> /s)	
19	bypass flow in the unsaturated zone	(mm/h)	
20	depth of water in river	(m)	
21	snow storage	(mm)	X
22	river flow at specified stations	(m <sup>3</sup> /s)	
23	flow in the unsaturated zone	(mm/h)	
24	water content at each uz-node	(-)	
25	total inflow to river from overland	(m <sup>3</sup> /s)	
26	total inflow to river from aquifers	(m <sup>3</sup> /s)	
27	total inflow to river from drainage	(m <sup>3</sup> /s)	
28	effective water content in unsaturated zone	(-)	
29	total irrigation	(m <sup>3</sup> /s)	
30	irrigation intake from the river	(m <sup>3</sup> /s)	
31	irrigation pumping from wells	(m <sup>3</sup> /s)	

Figure 17 Data types available.

## Plot types

The plot type determines the layout of the plot analogous to the line type in connection with time series plots. The available plot types can be listed by pressing the <display plot types> button on menu G.1.2.

The following plot types are available.

#### HORIZONTAL PLOTS (layer plots).

- 1 : Grid values. (numbers)
- 2 : Grid values. (colours)
- 3 : Contour plot. (lines)
- 4 : Contour plot. (coloured)
- 5 : Water level in rivers. One grid diagonal corresponds to 1 meter water in the river.
- 6 : Vector plot, horizontal flow. (datatype 16 and 18 only)

10-80 : Landmarks (digfiles). Plottypes less than 80 which are a multiple of 10 gives polygons with colours 1 to 8 (e.g. 20 gives a polygon with colour code 2). Remaining plottypes plots lines or points with colour 1-8 and thickness 0.1-0.8 mm. (e.g. 25 gives a line with colourcode 2 and thickness 0.5 mm).  
The same applies for plottypes greater than or equal to 80, but the colour is determined by value and the current scale.

#### CROSSSECTION PLOTS (plottype 100-103)

- 100 : Cross-section
- 101 : Potential head plotted in a cross-section
- 103 : Darcy flow vectors in cross-section [m/day]

#### SPECIAL FEATURES AND DATA MANIPULATION.

- +1000: Add to the data in the line above (plottype 1-4 only)
- +2000: Exclude model boundaries from plot (plottype 1-4 only)
- +5000: Don't plot the data (all plottypes)

Figure 18 Plot types available.





As for time series plots data type and plot type can be combined almost as desired. In principal horizontal 2D data can be plotted in four different ways namely as grid values presented as colours or numbers, or as contours plotted as lines or colours. Figure 19 shows the four basic plot types:

1. values
2. colours
3. contour lines
4. coloured contour

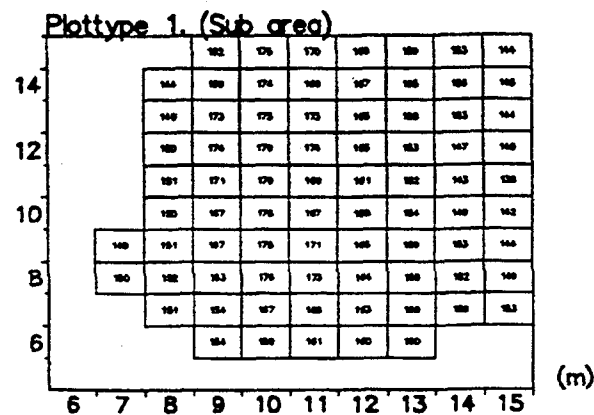


Figure 19 Surface topography (data type 60) plotted as plot type 1-4.

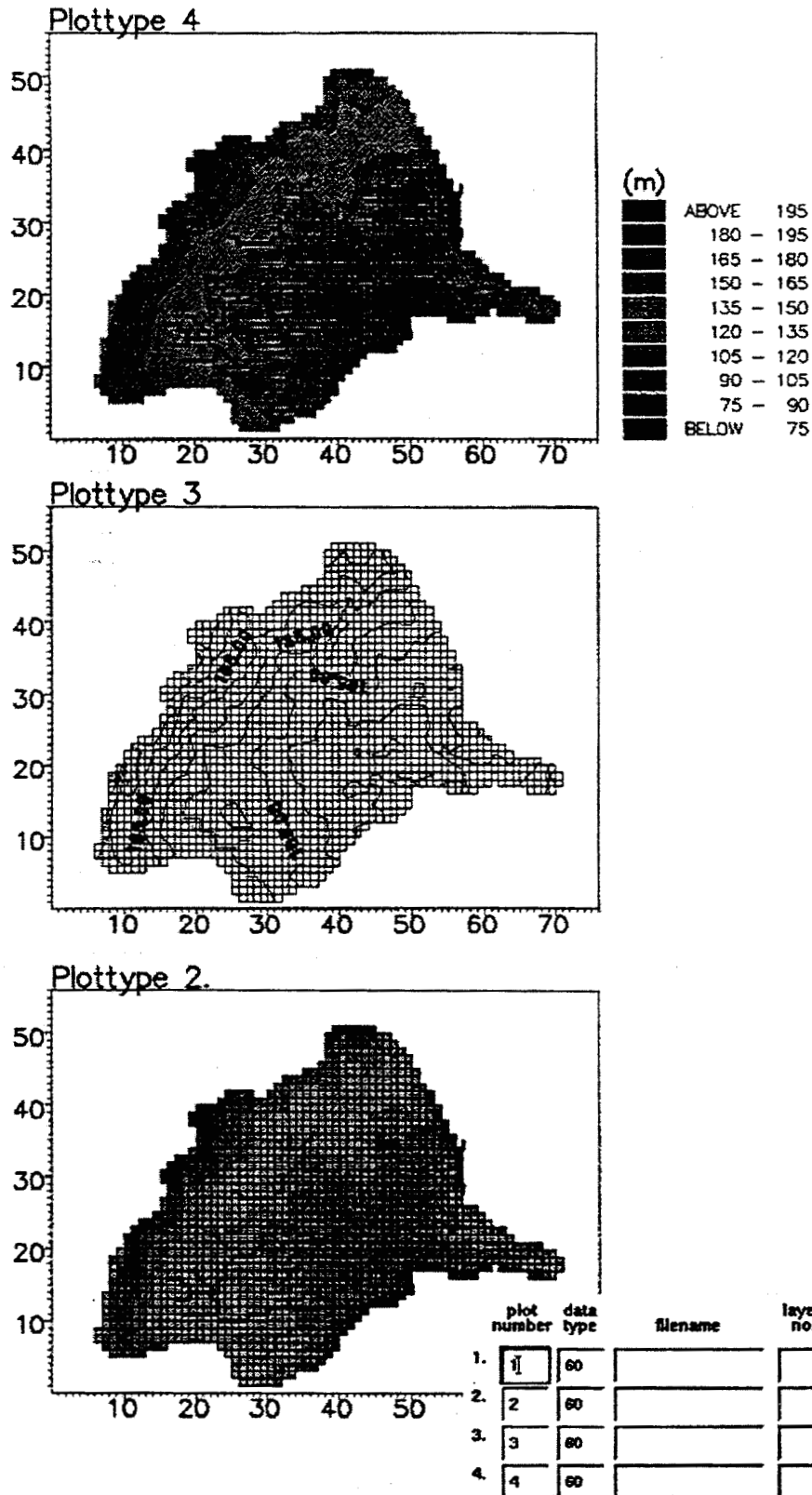


Figure 19 Surface topography (data type 60) plotted as plot type 1-4 (cont.).



## Modifying the plot layout

Except plot no. 1 the four plots shown in Figure 19 applies default values for geometry, colours etc. The defaults can be changed by entering menu G.1.2.a - PLOT LAYOUT.

Figure 20 Menu G.1.2.a, Plot layout.

The Y-axis, data transformation and plot geometry is defined as for time series, thus it will not be further discussed in this section.

## Output date

If you want to plot simulation data e.g. potential head (data type 15) you must specify the desired output date. The <step dt> button will step to the next or the previous storing time step for the actual data type. You can also just specify the desired output date directly.



## Plot area

The plot area will be initialised to the entire model area when an input data file is loaded. If only a specific part of the area should be plotted the X and Y co-ordinates can be modified as desired. The plot area is also used to specify the location of cross-sections in the grid network. In this case Xbeg, Xend and Ybeg, Yend will be substituted with point co-ordinates X1, Y1 and X2, Y2. The field <No. of sections> defines the number of subsections in a cross-section. Each extra section will need specification of one extra co-ordinate. You can step between the sections (co-ordinates) by clicking the arrow button.

## Colours and classes

The colour scale that is applied to plot type 2 and 4 in Figure 19 is a standard UNIRAS topographical colour scale. Other standard colour scales can be selected by specifying the number of the desired colour scale. The available colour scales can be listed by pressing the <display colour scales> button on menu G.1.2.a.

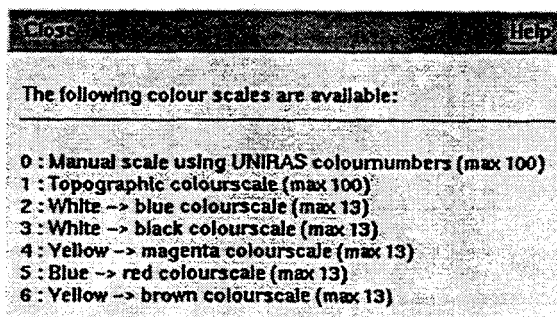


Figure 21 Colour scales available.

If a standard colour scale is applied (scale 1-6) the colour codes cannot be modified. User defined colour scales can be defined by specifying colour scale 0 (manual) and subsequently specify any desired colour code. The number of colours can be specified for any colour scale. The colours will be attached to a certain data value defined by the colour classes (data intervals). The number of classes is always one less than the number of colours. If all colour classes are zero the MSHE.GP performs automatic scaling depending on the number of classes and the maximum and minimum values for the actual data set. Below is given an example of a user defined colour scale using mixed UNIRAS colours.



colour scale : 0

no. of colours : 5

actual number	colour code	colour class
1	-010101	100.0 (m)
2	-030303	120.0
3	-050505	140.0
4	-060606	160.0
5	-070707	

If the surface topography shown in Figure 19 is plotted using this user defined colour scale it will appear as shown in Figure 22.

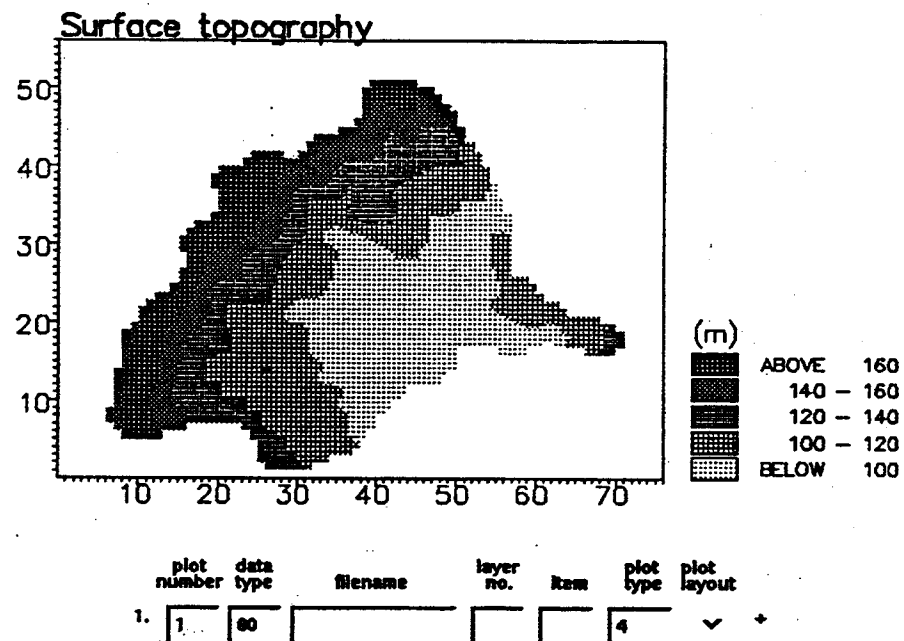


Figure 22. Surface topography using a user-defined colour scale.

### Water depth in the river

The water depth in the river system (data type 20) at a specified output time can be plotted as shown in Figure 23.

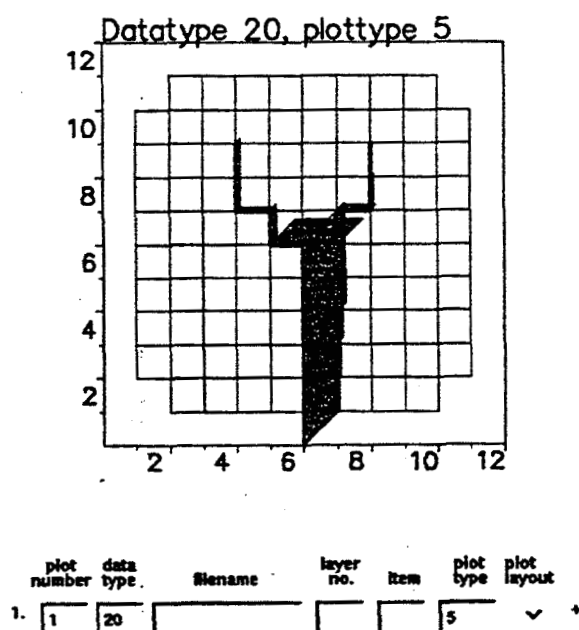


Figure 23 Water depth in the river system.

The water depth is plotted as a number of parallelograms having an angle of 45 degrees related to the orientation of the grid network. The height of each parallelogram defines the water depth at the actual location in the river. One grid diagonal corresponds to a water depth equal to the multiplication factor (metres). In Figure 23 is applied a multiplication factor equal to one implying that one grid diagonal corresponds to one meter, thus the water depth in the river is between 0 and approximately 1.2 metres at the selected output time.

### Vector plots

Simulated flow velocities in the saturated zone (data type 16) or on the ground surface (data type 18) can be plotted as vectors in the grid network. Figure 24 shows a horizontal plot of the groundwater flow in a selected layer at a certain output date.

The length of the "unit arrow" can be scaled using the multiplication factor on menu G.1.2.a. For the vector plot in Figure 24 a factor equal to 0.05 has been applied, implying that the length of the unitvector corresponds to 0.05 m/day. The colour of the vectors can be selected by specifying the desired colour code in menu G.1.2.a.

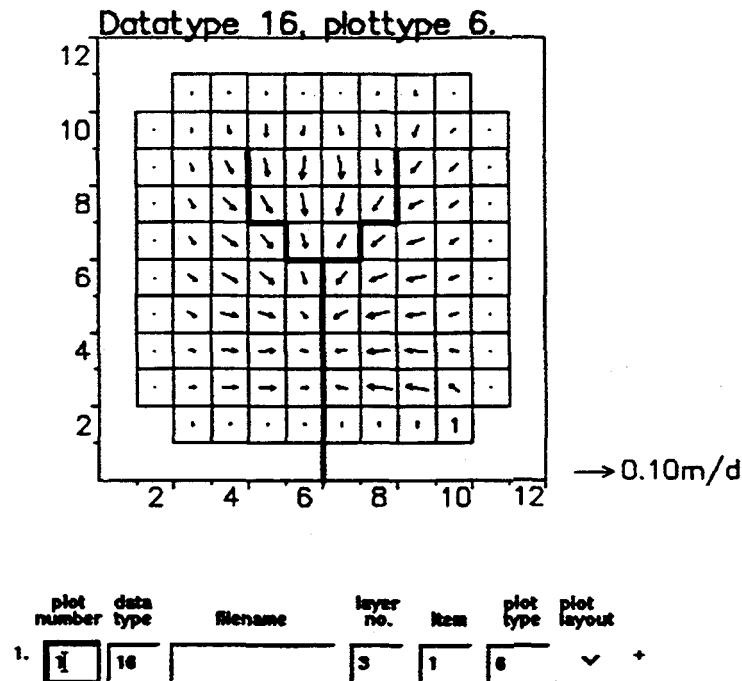


Figure 24 Vector plot of ground water flow velocities.

### Location of head observation wells and groundwater abstractions

The location of head observation- or groundwater abstraction wells, and the actual value at a specified output date can be plotted in a horizontal grid plot (for plot type 1-4) as shown in Figure 25. The potential head observation wells are plotted as a coloured circle where the colour represents the actual observed head. If the potential head observation is plotted overlaying the corresponding simulated head (only plot type 2 and 4) the same colour scale is applied to observed and simulated data, which is a convenient feature for comparison of observed and simulated data. Groundwater abstractions are plotted as a point attached to the actual value.







### AD- and PT-results

When plotting spatially distributed data from transport simulations results (AD or PT simulation results) attention must be paid to some of the specifications.

An input file or a result file must be selected in menu G.1 for the MSHE.GD to know about the grid dimensions etc.

Similar to plotting time series, specification of the transport simulation result file is needed when plotting transport data (data types 41-46).

This is done in menu G.1.2. If concentrations from an AD-simulation (data type 41) is to be plotted the species number must be specified under item.

Specification of the output date must be done directly in menu G.1.2.a, there is no default date like the simulation start, nor is it possible to use the step dt arrows. Results at the output date will be plotted, but if the specified date is between two output dates the succeeding date will be used.

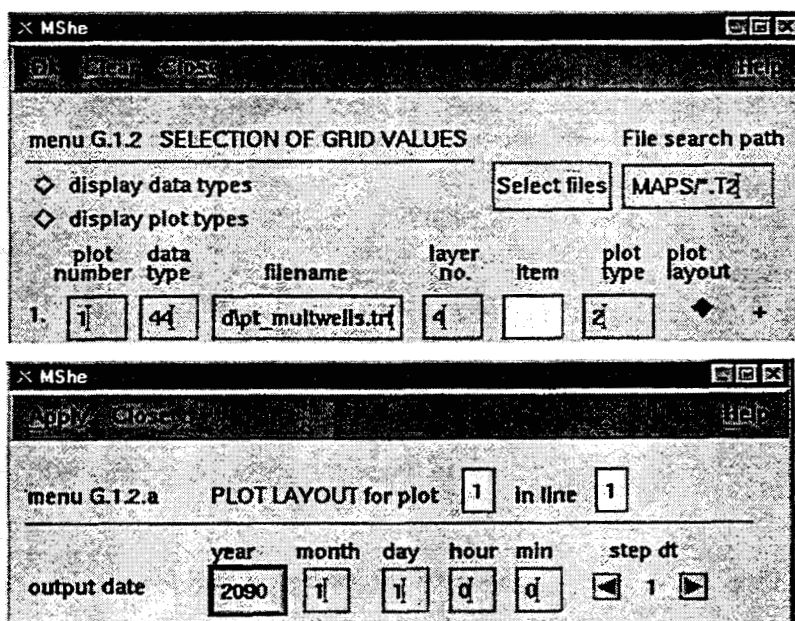
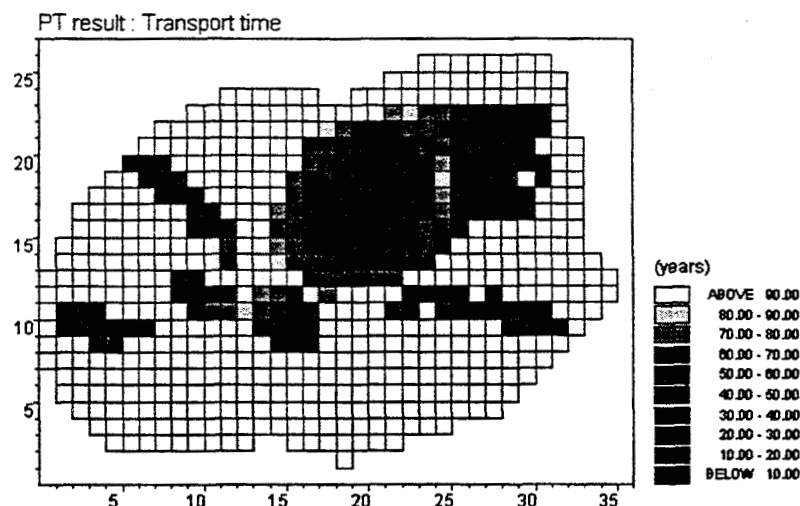


Figure 26 Transport time for the water, i.e. time for the particles to reach the wells, and the menu parts to pay attention.

### Plotting cross-sections

Cross-sections can only be plotted for a few data types namely for the hydrogeological parameters (data type 61 - 65) and transport simulation parameters (data type 41-46) and for simulated potential heads and flow velocities, as plot type 100, 100, 101 and 103 respectively. Below is shown some examples of cross-section plots.

Figure 27 shows a cross-section of the computational layers (data type 61). The cross-section consists of two subsections defined by three grid co-ordinates (20,10) , (45,20) and (50,40). The applied colours and the text legends as well as the grid co-ordinates are specified on



menu G.1.2.a. If no colours are specified on menu G.1.2.a MSHE.GP applies default colours for each layer. The layer number that is specified for a cross-section is the last layer to be included in the plot. A cross-section will always start from the top layer (layer 1).

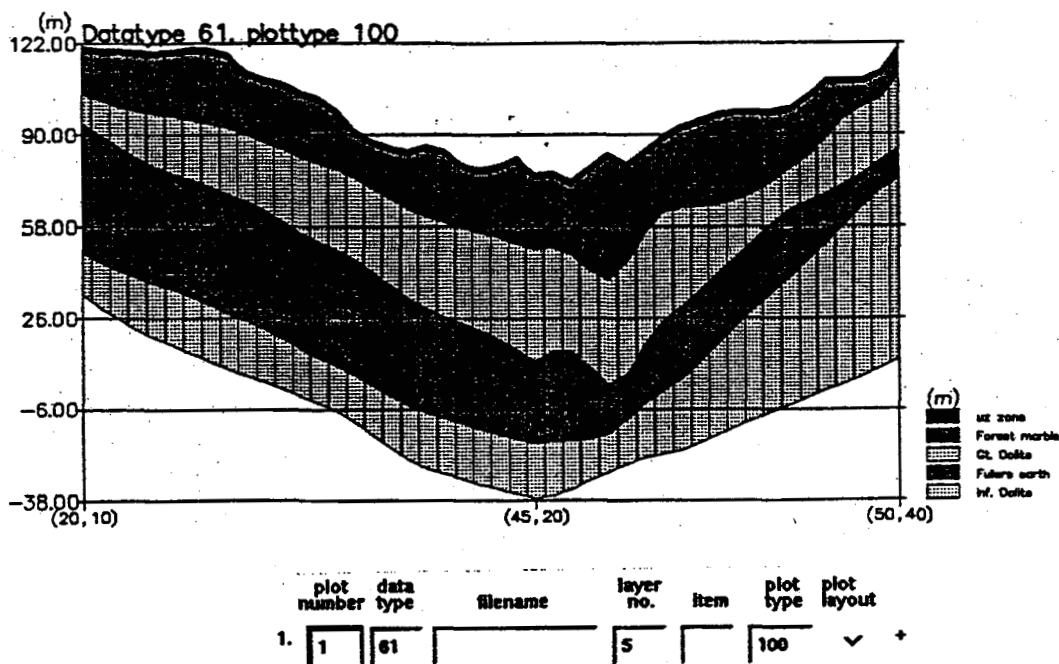


Figure 27 Cross-section of computational layers.

Figure 28 shows a cross-section of horizontal hydraulic conductivities (data type 62) overlaid by simulated groundwater flows (data type 16) plotted as vectors (plot type 102). Any default or manual colour scale can be used for the hydraulic conductivity and the arrows can be scaled and coloured as for horizontal plots described earlier in this section.

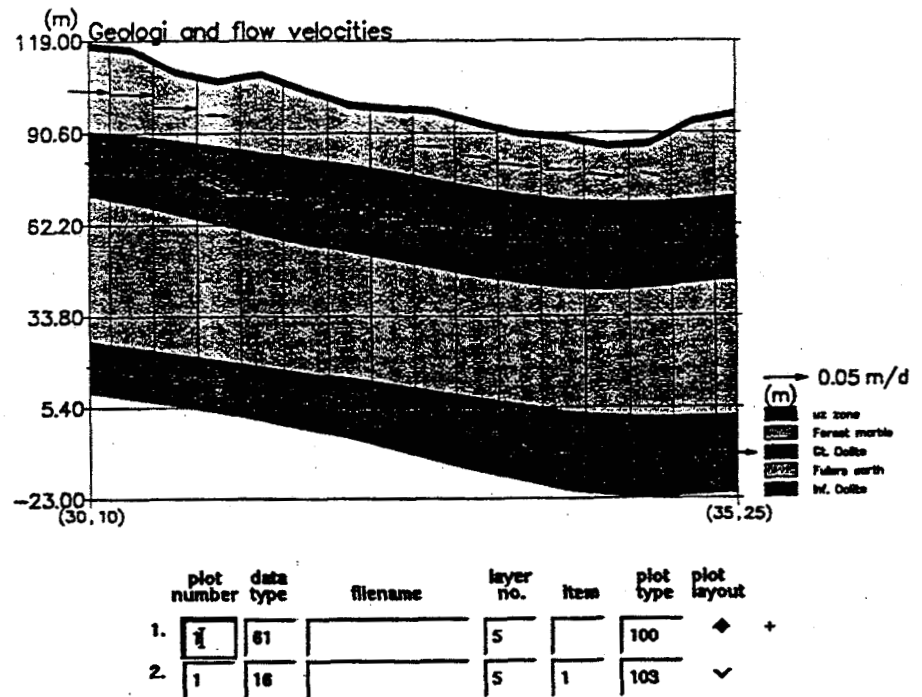
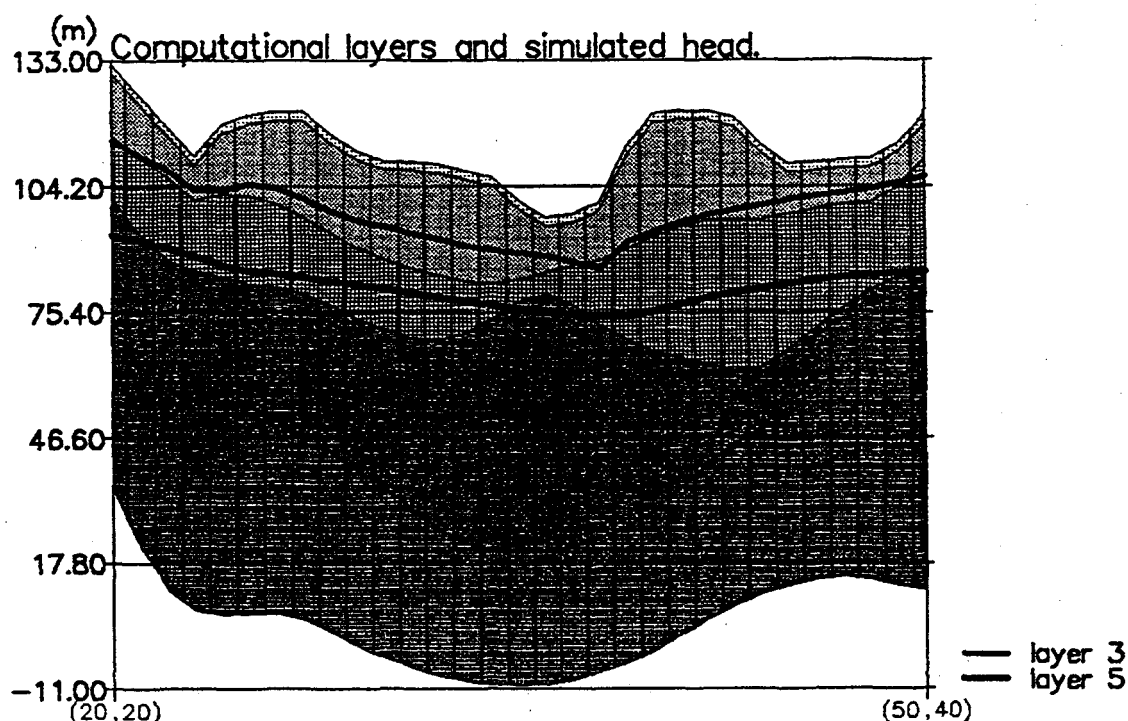


Figure 28 Cross-section of horizontal hydraulic conductivity and flow velocity in the saturated zone.

Figure 29 shows a cross-section of the computational layers together with the simulated potential head in layer number three. The line that represents the potential head can be plotted in different colours and thicknesses by specifying a colour code on menu G.1.2.a, representing a line type. The line types used for lines in time series plots (see Figure 4) can be applied, though only line types 0-335 are valid.



	plot number	data type	filename	layer no.	item	plot type	plot layout
1.	1	61		5	1	100	◆ +
2.	1	15		5	1	101	▼
3.	1	15		3	1	101	▼

Figure 29 Cross-section of computational layers overlaid by the simulated potential head.

Figure 30 shows a cross-sections of the simulated transport time of the water, together with the menu specifications for the plot. Plotting cross-sections of transport simulation results differs from plotting the data as 2D horizontal data only in the specification of the plot type, which must be 100.

Again it is necessary to select the transport simulation result file in menu G.1.2, and the output date in menu G.1.2.a. Still the arrow buttons for step dt does not work.

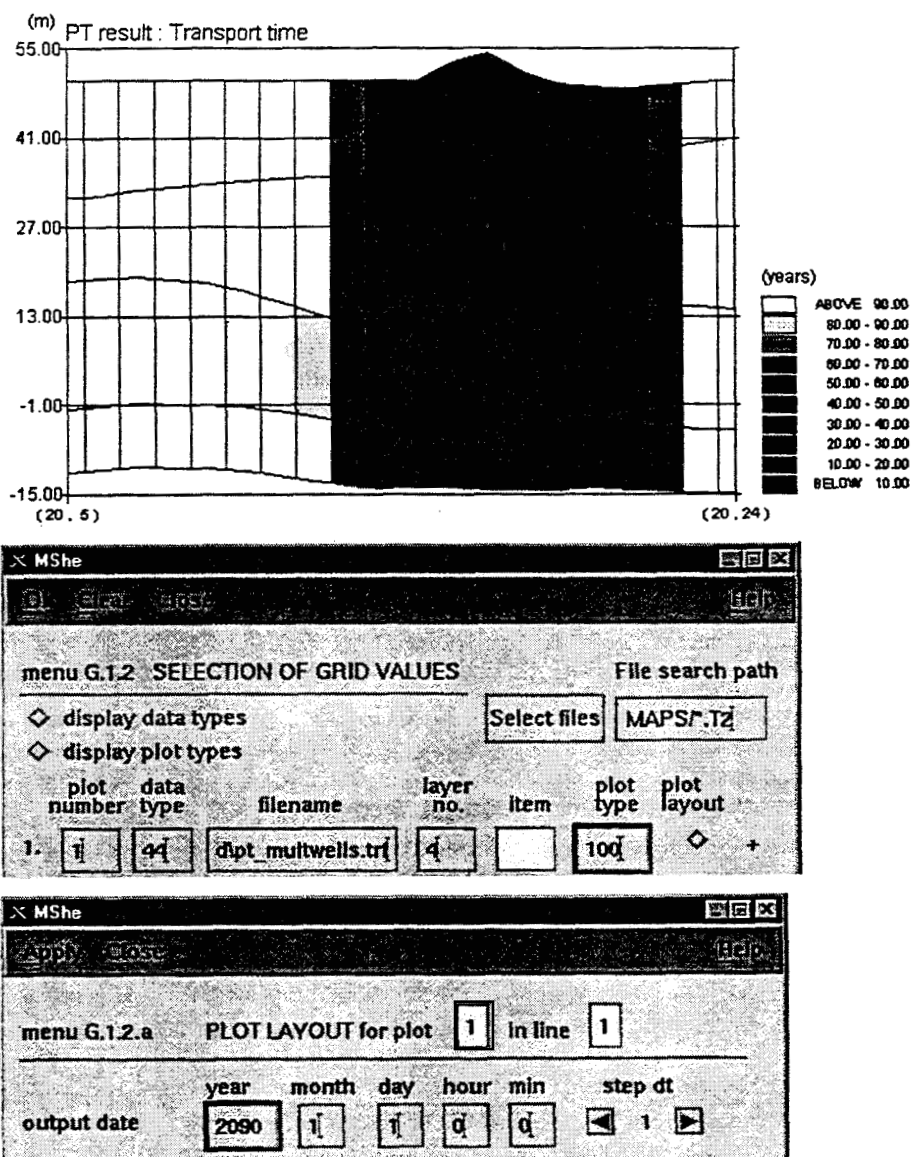


Figure 30 Cross-section of transport time for the water, i.e. time for the particles to reach the wells, and the menu parts to pay attention.

### Plotting digitised data and landmarks

The MSHE.GP includes facilities for plotting data stored as data file type 31 and 32 (see the **Data File Format** section in this manual). This facility can be used to plot raw digitised data as topographical contours or digitised polygons. Typically the data will be the output from a digitising program, but since the applied data file format is a simple x,y,z co-ordinate based format stored on ASCII files the data may be prepared or modified manually using a text editor. This allows the user to produce any geometrical figure that can be represented as a point, a line or a polygon.



## Plotting points, lines and polygons

The data can be plotted as data type 90, 91 or 92 (see Figure 17) which in combination with the plot type determines the layout of the plot. The plot type can be 10-99 and is considered as integer codes defining the applied colour and the thickness of a line or the diameter of a point. The first integer determines the colour (default colour 1-9) and the second code the thickness or diameter.

For instance plot type 25 would give the colour 2 (red) and the thickness or diameter code 5 (thickness 0.5 mm , diameter 5 mm).

Whether the data are plotted as lines or points depends on the code value specified as the first value in the data lines in the input data file. This code value can be either 1,2 or 3, where :

- 1 : is start of new line or polygon;
- 2 : is a point on a line or a polygon;
- 3 : is end of line or polygon.

A line contains at least two points (code 1 and 3) and a polygon at least three points (code 1,2 and 3). If only code three is present the data are considered to be a point. In Figure 33 are given some examples that illustrates the basic features. The data files that are plotted are shown in Figure 31 and Figure 32.

```
FILETYPE DATATYPE Verno: 31 0 501
TEXTLINE          : digitised triangle
UTM XYUNIT        : 0 1
1  1.0  1.0  10.0
2  2.0  1.0  10.0
3  1.5  2.0  10.0
```

Figure 31 Input data file *triangle.dig* including three points that can be plotted as line segments or a polygon (triangle).

```
FILETYPE DATATYPE Verno: 31 0 501
TEXTLINE          : digitised points with text
labels.
UTM XYUNIT        : 0 1
3  1.5  0.5  10.0  'label 1'
3  0.5  2.5  10.0  'label 2'
```

Figure 32 Input data file *labels.dig* including two points and labels that can be plotted as points (data type 90,91) or labels (data type 92).

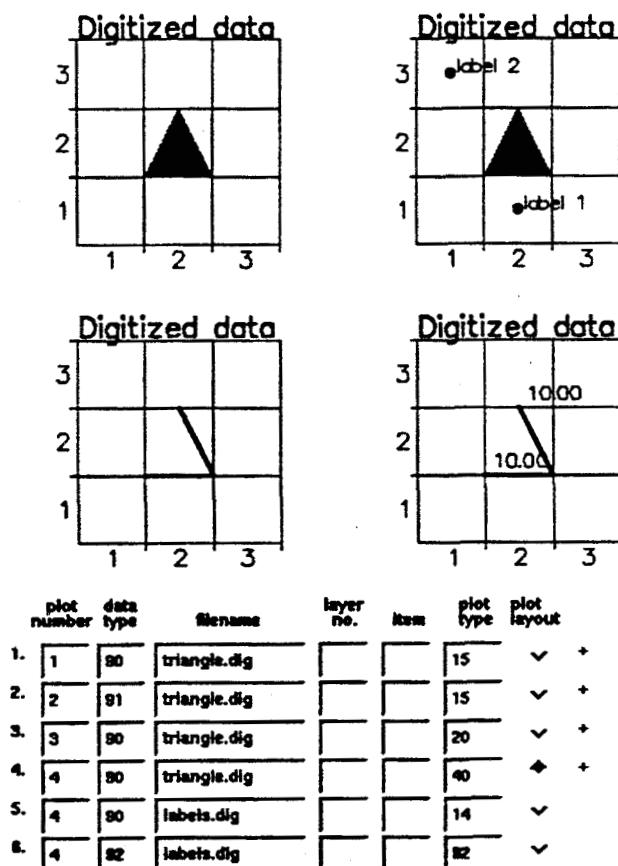


Figure 33 The input data files *triangle.dig* and *labels.dig* plotted as line segments, polygons, points and labels.

As described above the colour and thickness or diameter is determined by the specified plot type. If another colour than the default colours (1-9) is desired any colour code can be specified in menu G.1.2.a, overwriting the one specified by the plot type. The figures below illustrates a number of possible facilities plotting raw digitised data and generating landmarks as roads, cities, rivers etc.



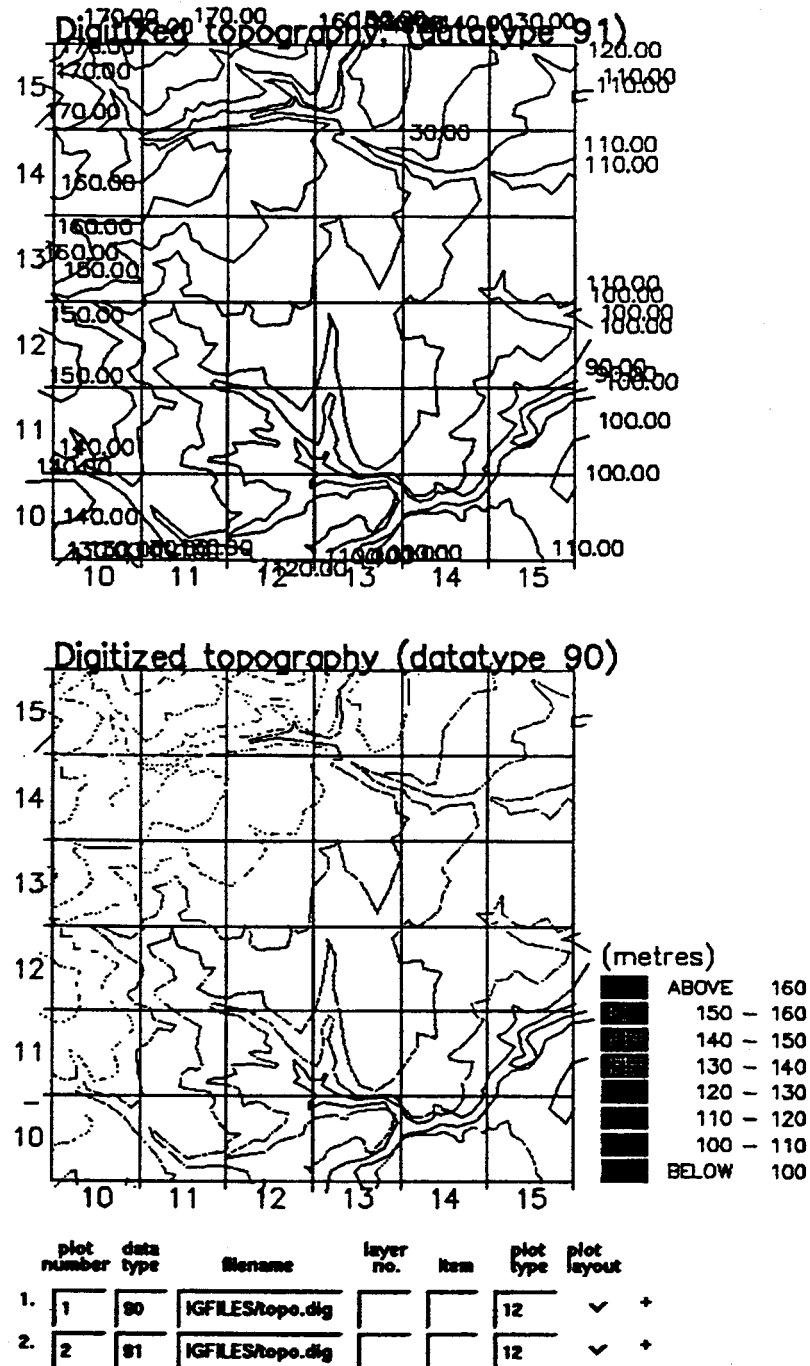
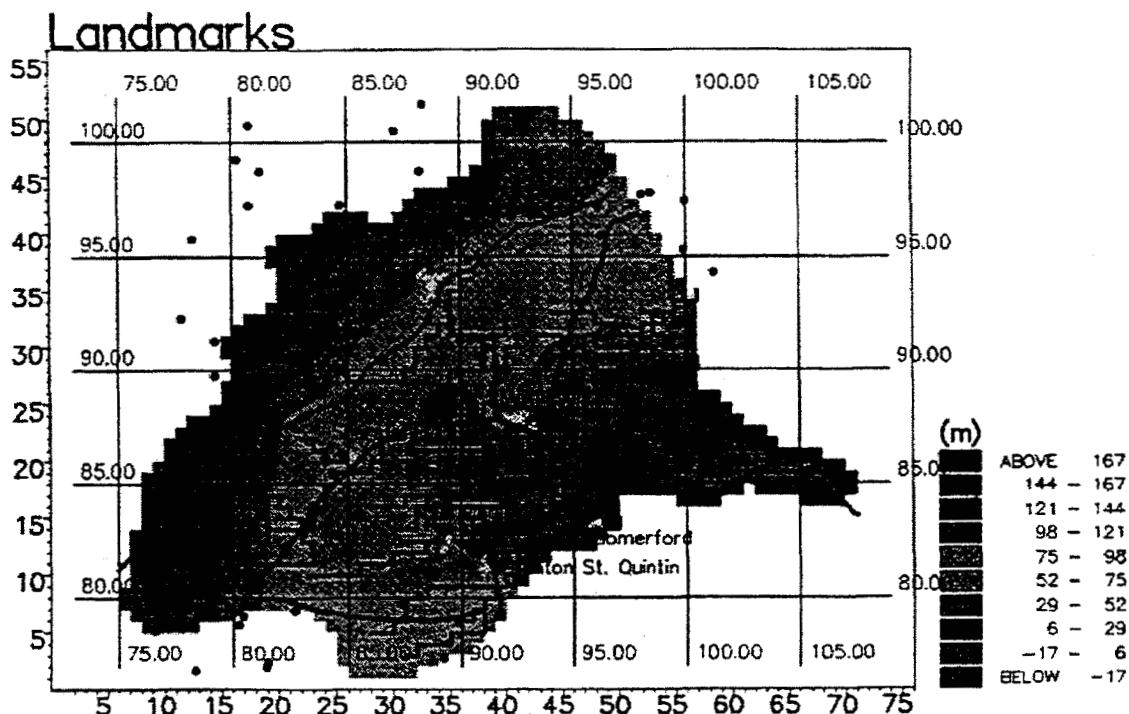


Figure 34 Digitised surface topography plotted as contours using data type 90 and 91, giving a data value attached to each contour or a colour scale representing the actual elevation.



	plot number	data type	filename	layer no.	item	plot type	plot layout
1.	1	61		3	1	4	◆ +
2.	1	80	landmk/cities.dig			10	✓
3.	1	80	landmk/river.plot			51	✓
4.	1	91	landmk/gridref.dig			42	✓
5.	1	80	landmk/road.dig			25	✓
6.	1	82	landmk/citybd.dig			22	✓
7.	1	80	landmk/len94.xyz			63	✓

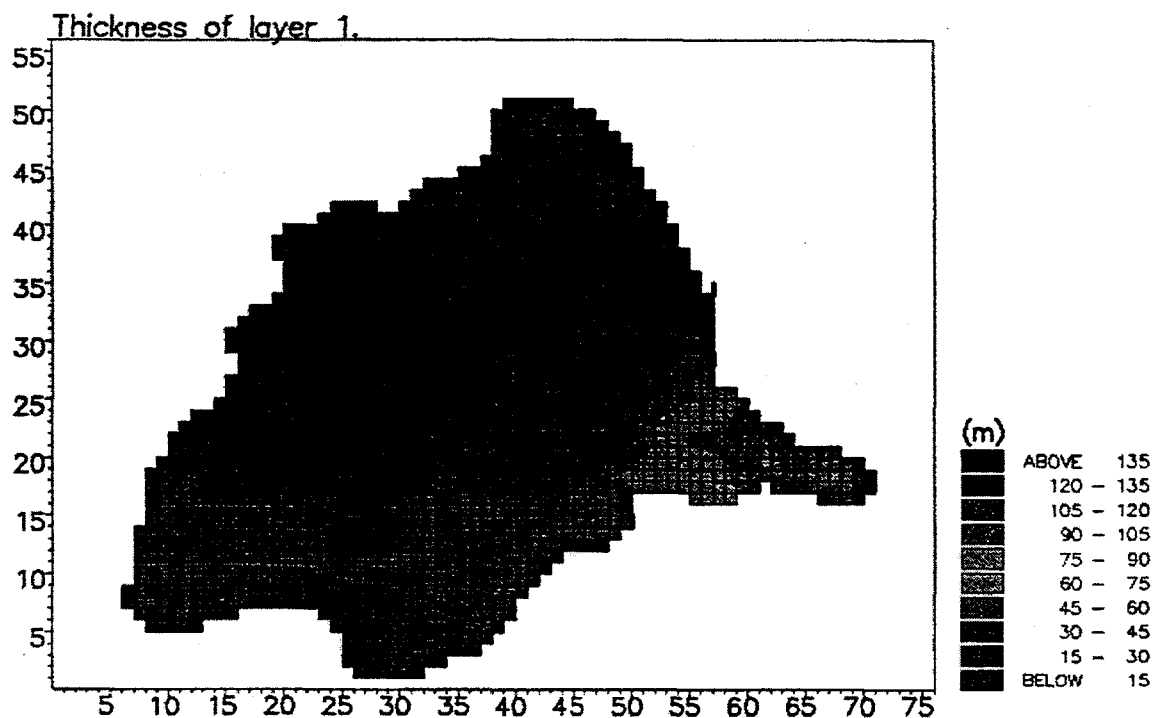
Figure 35 Landmarks plotted as digitised data.

### Special plotting features

Horizontal 2D matrix data can, similar to time series, be manipulated before plotting. By adding 1000 to the plot type two data sets can be added (see Figure 18) or subtracted using a negative multiplication factor. In Figure 36 is given an example where two layer boundaries are subtracted in order to calculate and plot the thickness of a layer. The layer thickness is plotted as a coloured contour plot (plot type 4). Plot type 5004 in specification line 1 implies that the data set should be read but not plotted. Plot type 1004 in specification line 2 implies that the data set should be added to the data set in the specification line defined above. The multiplication factor for specification line 2 (in menu G.1.2.a) is specified equal to -1.0 implying that the data set



specified in line 2 will be subtracted from the data set specified in line 1.



	plot number	data type	filename	layer no.	item	plot type	plot layout
1.	1	61		1		5004	✓
2.	1	61		2		1004	✓ +

Figure 36 Thickness of layer number one.

If 2000 is added to the plot type the model boundaries will be excluded from the plot. This is a useful option if plotting data where no (or dummy data) is available on the model boundary. For instance if plotting simulated potential head (data type 15) in a layer using zero flux boundary condition, hence no potential head is calculated on the boundary in this case.

## 1.4 Animation of MIKE SHE Results

The MSHE.GP includes facilities to generate videos by animation of simulation results. This is a very useful option in order to get a better understanding of the dynamics of the simulated system, as well an excellent way of demonstrating results for a client or at a conference.

Before describing how to animate the results the following terms should be defined:



**video file** : A file containing the animation. The data file must have the suffix **.vdo**.

**video player**: A program that reads a video file and display the video.

### Video file generation

The videos are generated by selecting the **video driver** device on menu G.1. When running the MSHE.GP a number of plots will be generated and written to the data file **video.vdo**. What the video player basically does is to read the video file and display the plots quickly after each other, more than 20 pictures per. second, generating an animation effect. One picture is generated for each timestep **<dtvideo>** within the period defined by start data and end date as specified on menu G.1. When producing videos it is recommended to prepare the plot on the monitor and not select the video device until the plot layout is as desired. You can produce videos of any plot that can be produced with the MSHE.GP including time series, gridplots, landmarks etc.

As mentioned above the animation works by showing pictures quickly after each other. Therefore it is important, in order to produce good videos, to observe that the video timestep (**dtvideo**) reflects the timescale of the simulated processes, i.e. there should not be too large changes from one picture to another. On the other hand you should also observe that a MIKE SHE result file may include several thousands timesteps, thus it will be very CPU consuming to produce one picture for each timestep as well as the video file will become very large. At present the MSHE.GP always names the video files **video.vdo**. This implies that you should rename the video file before generating a new one. Please observe that suffix should always be **.vdo**. Whenever a video is generated the MSHE.GP writes a data file **video.log** containing relevant execution information as errors or warnings. Among other things this data file also contains the size of each picture in bytes which can be a useful information in order to calculate the size of a video file before producing a video for a long simulation period.

### Video playing

The video player can run the animation in either automatic mode or in single step mode. In single step mode the video player displays one picture whenever a key is pressed on the keyboard. You can enter and exit single step mode by pressing the 's' key.



You can dump a picture from the animation into a .PCX data file by striking the 'd' key. PCX is a very common picture format in the DOS world. Most drawing programs can import PCX data files, e.g. Paintbrush in Microsoft Windows. You can use this to make slide shows or elaborate a picture before using it in a report.

The escape key exits the animation.

The video can be displayed both on DOS and UNIX (X-windows) computers. The names of the players are **v12play.exe** for DOS and **xplay** for X-window. Both players (programs) takes the following options:

- i name    the name of the video file, without the .vdo extension. If you do not use this option the program searches for **video.vdo** file;
- t num     slows down the animation by waiting **num** ticks between each picture;
- l num     rerun the video **num** times. By default the animation is shown 10 times.

### Special for v12play.exe

This video player requires a VGA video card, which is the most common card in modern PC's. As the video player utilises VGA mode 12h to show the graphics, there are two limitations you should observe when generating the video file. There are a maximum of 16 different colours, including black and white. The other limitation is that a single picture must not be bigger than 50,000 bytes. This can be checked in the **video.log** file.



# **MIKE SHE PP – User Manual**

## **Water Balance Utility**





## CONTENTS

<b>1</b>	<b>WATER BALANCE UTILITY .....</b>	<b>1</b>
1.1	Macro File.....	2
1.1.1	Input to MIKE SHE water balance extraction program .....	2
1.1.2	Input to MIKE SHE water balance post-processing program.....	3
1.1.3	Example of macro file .....	4
1.2	Output Formats .....	4
1.2.1	Table format .....	5
1.2.2	T0 format.....	6
1.2.3	T2 format .....	6
1.2.4	Chart format .....	7
1.3	Storage Requirement in the Flow Result File .....	8
1.4	Configuration File .....	9
1.5	Water Balance Items .....	10
	Snowmelt Terms (SM).....	11
	Canopy Interception Terms (CI).....	11
	Overland Terms (OL).....	12
	River Terms (RIV).....	13
	Unsaturated Zone Terms (UZ).....	13
	Saturated Zone Terms (SZ).....	14
1.6	Running the Water Balance Utility .....	14
1.6.1	Calculate the water balance .....	15
1.6.2	Illustrate the water balance in a chart.....	15

194







## 1 WATER BALANCE UTILITY

The MIKE SHE water balance utility is a post-processing tool, which serve the following purposes:

- A clear unambiguous summary of flow and storage changes for the individual model components as well as for the entire model.
- Analysis of errors occurring in the model due to input data or numerical instabilities.

The water balance utility enables the user to present results for the entire model area or a sub-catchment area (down to single columns).

The MIKE SHE water balance calculation consists of two steps:

1. Execution of the water balance extraction program (mshe\_wbl\_ex.exe). The extraction program calculates the "gross" water balance for the total catchment or one or more sub-catchments.
2. Execution of the water balance post-processing program (mshe\_wbl\_post.exe). The post-processing program extracts user defined water balance data.

Figure 1 shows a flow chart for the water balance utility.

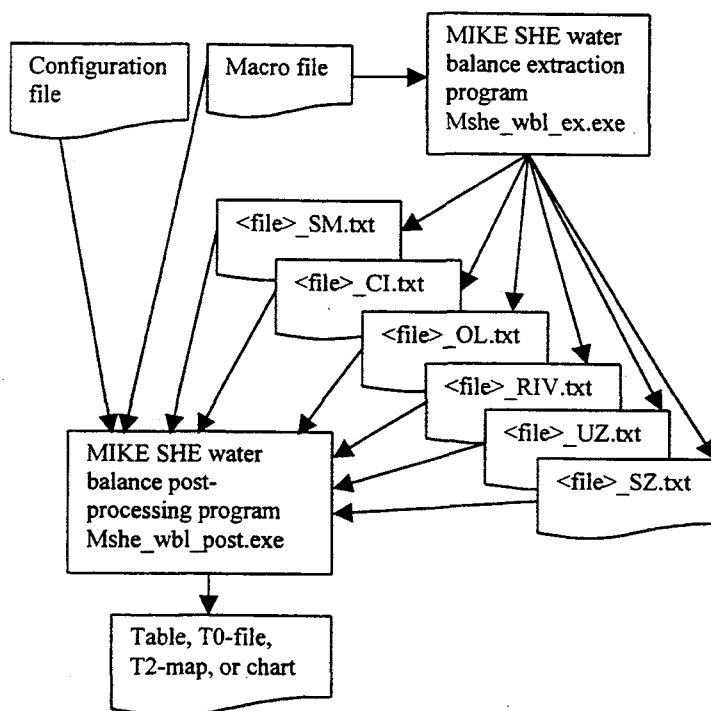


Figure 1 Flow-chart for the water balance utility

## 1.1 Macro File

The specifications for the extraction and post-processing program are given in a macro file.

### 1.1.1 Input to MIKE SHE water balance extraction program

Result file	Name of flow result file – extension frf required but no path required.
Wbl- discretisation	Determine the region(s) for which water balance are performed: 1=catchment, 2=sub-catchment, 3=single cell in total catchment, 4=single cell in areas with sub-catchment grid code=1
Sub-catchment file	Definition of water balance sub-catchments. Only required for wbl-discretisation option 2 and 4. The sub-catchment file (T2-map) should contain zeros in areas where no water balance is wanted and positive number in areas where water balance is wanted. A sub-catchment is defined as a region with same code. The sub-catchment file can contain an arbitrary number



of sub-catchments. It is recommend that the sub-catchments are numbered from 1 to the number of sub-catchments.

Gross list filename      "base name" for gross list files. Component gross list files are named <filename>\_sm.txt, <filename>\_ci.txt, etc.

When any of the items in Section 1.1.1 are changed both the MIKE SHE water balance extraction program and the MIKE SHE water balance post-processing program has to be executed.

### 1.1.2    *Input to MIKE SHE water balance post-processing program*

Start time	Start date and time for the post-processing of the gross lists
End time	End date and time for the post-processing of the gross lists
Output time step	Time step used in post-processing of the gross lists. The output time step is adjusted to the maximum storing time step if it exceeds the specified output time step.
Output type	1: incremental water balance, 2: accumulated water balance
Water balance type	(SM, CI, OL, RIV, UZ, SZ, TOTAL, TOTAL ERROR, etc.) The water balance types are defined in the configuration file
Output format	1: Table, 2: T0-file, 3: T2 map, 4: Table for Chart Program, see Section 1.2
Explode layers	(T/F) water balance showed for all layers when TRUE
Output layer	If the output layer is different from 0 a water balance for the specified layer will be performed
Output location	(ix, iy, isub) Sub-catchment for which water balance is wanted. The sub-catchment is either given as grid coordinates or as a sub-catchment number. If ix and iy equals 0 then isub will be used as sub-catchment. Otherwise the program will determine the sub-catchment form the grid co-ordinates.
Output file name	Name of output file produced by the post-processing program
Default config file	(T/F) when true a default configuration file is used. The file is located as %shedir%\bin\config.wbl, otherwise the configuration file has to be specified



Configuration file      Name of the configuration file for the post-processing program (if Default config file = FALSE), see Section 0

When any of the items in Section 1.1.1 or in Section 1.1.2 are changed the MIKE SHE water balance post-processing program has to be re-executed.

### 1.1.3 Example of macro file

FILETYPE DATATYPE VERN0: 10003      0      501

-----  
Setup for MIKE SHE extraction program - run mshe\_wbl\_ex.exe after editing  
-----

Result file            : wle1.frf  
Wbl- discretisation : 2  
Sub-catchment file : maps\sub-cat.T2  
Gross list filename : wbl\gross  
-----

Setup for MIKE SHE post processing program - run mshe\_wbl\_post.exe after editing  
-----

Start time            : 1990 6 1 0 0  
End time             : 1990 7 12 0 0  
Output time step    : 4  
Output type          : 1  
Water balance type : SZ  
Output format        : 1  
Explode layers       : F  
Output layer          : 0  
Output location      : 0 0 3  
Output file name     : wbl\szwbl.txt  
Default config file : F  
Configuration file   : c:\shedata\config.wbl

## 1.2 Output Formats

The post-processing program can produce output in four different formats: Table format, T0 format, T2 format and Chart format. All formats except for the T2 format present a water balance for one sub-catchment over a given period of time. The T2 format presents a single water balance group for all sub-catchments.

All results are presented in units of millimetres of water per area. If for an example we look at abstraction presented in a table format with an output time step of 7 days the result will be the accumulated abstraction over 7 days normalised by the area of the observed sub-catchment. Results presented in map or chart format are accumulated over the entire output period specified in the macro file. Results can be converted to m<sup>3</sup> of water by multiplying the result by "the number of cells in the sub-catchment \* cell area/1000"



## 1.2.1 Table format

The table format is a tab-separated ASCII file that can be analysed by any text editor. Table 1 shows an example of the table format.

*Table 1 An example of the table format of the water balance utility. The table shows the accumulated error of layer 1 to 6 for two different time steps distributed with respect to canopy interception (CI), snow melt (SM), overland (OL), river (RIV), unsaturated zone (UZ), saturated zone (SZ), and finally the total error for a selected catchment.*

```
FILETYPE DATATYPE VERN0: 10006 0 544
ERROR water balance      - accumulated      - layer 1 to 6      :      1991 1 1 0 0      1994 1 1 0 0
CI OL RIV UZ SZ
Bjerge River Catchment
DHI
CI ERROR      SM ERROR      OL ERROR      RIV ERROR      UZ ERROR      SZ ERROR      TOTAL ERROR
1991 1 26 0 0      .3250502E-03      .0000000      -.1609759E-03      -.1339663      -4.056832      -.4807981E-01      -4.198059
1991 1 26 0 0      .0000000      .0000000      .0000000      .0000000      .0000000      -.7424196E-02      -.1380522E-04
1991 1 26 0 0      .0000000      .0000000      .0000000      .0000000      .0000000      -.1380522E-04      .1757028E-05
1991 1 26 0 0      .0000000      .0000000      .0000000      .0000000      .0000000      .1757028E-05      -.1669177E-04
1991 1 26 0 0      .0000000      .0000000      .0000000      .0000000      .0000000      -.1669177E-04      -.6024096E-05
1991 1 26 0 0      .0000000      .0000000      .0000000      .0000000      .0000000      -.6024096E-05      -.5553877E-01
1991 2 25 0 0      -.6327183E-03      .0000000      .5775230E-03      -.2034599      -6.691113      -.6135943E-01      -6.892560
1991 2 25 0 0      .0000000      .0000000      .0000000      .0000000      .0000000      .2069026E-02      -.9036145E-05
1991 2 25 0 0      .0000000      .0000000      .0000000      .0000000      .0000000      -.9036145E-05      -.1066767E-04
1991 2 25 0 0      .0000000      .0000000      .0000000      .0000000      .0000000      -.1066767E-04      -.1418173E-04
1991 2 25 0 0      .0000000      .0000000      .0000000      .0000000      .0000000      -.1418173E-04      .2761043E-05
1991 2 25 0 0      .0000000      .0000000      .0000000      .0000000      .0000000      .2761043E-05      -.5932153E-01
```

The format is suitable for viewing and editing in a spread sheet program. The table presents water balances over a given period of time with specified time steps. The water balance can be incremental or accumulated. If SZ is included in the water balance either the individual layers can be presented as additional rows for each time entry or the water balance can be presented for all layers in one row.

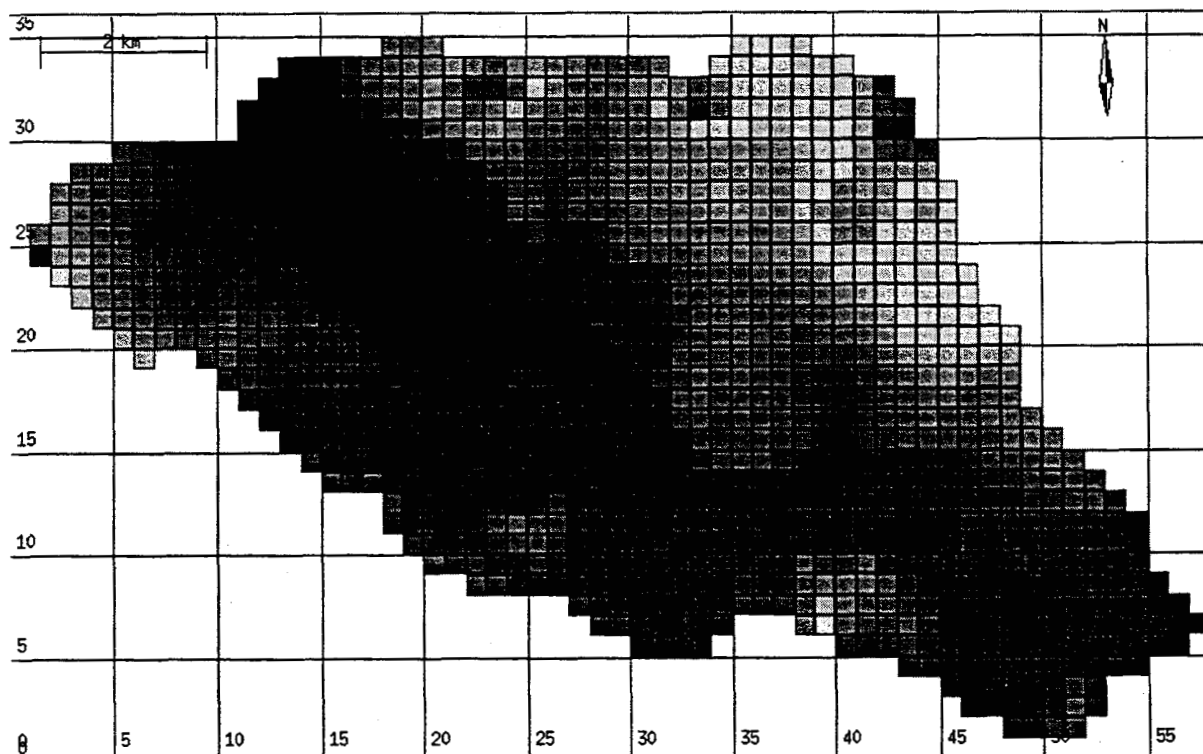


### 1.2.2 T0 format

The T0 format is at the present stage not implemented.

### 1.2.3 T2 format

The T2 output option is designed for viewing the distribution of a single water balance group. Specifically, the option is used for analysing the distribution of errors. The distribution of errors is presented in sub-catchments. If the water balance is calculated in all cells it will be possible to present the error (or any other parameter) in all cells. The presented values are accumulated values from start date to end date. An example of the T2 output is given in Figure 2.



**Figure 2** An example of the T2 output map. The map shows the distribution of the error of the unsaturated zone component. As this error was significantly influencing the total error according to Table 1 a further investigation of the error source can be done by means of the T2 map.

The T2-files can be edited from the MIKE SHE 2D Graphical Editor, see the MIKE SHE User Guide on Pre- and Post- processing.



### 1.2.4 Chart format

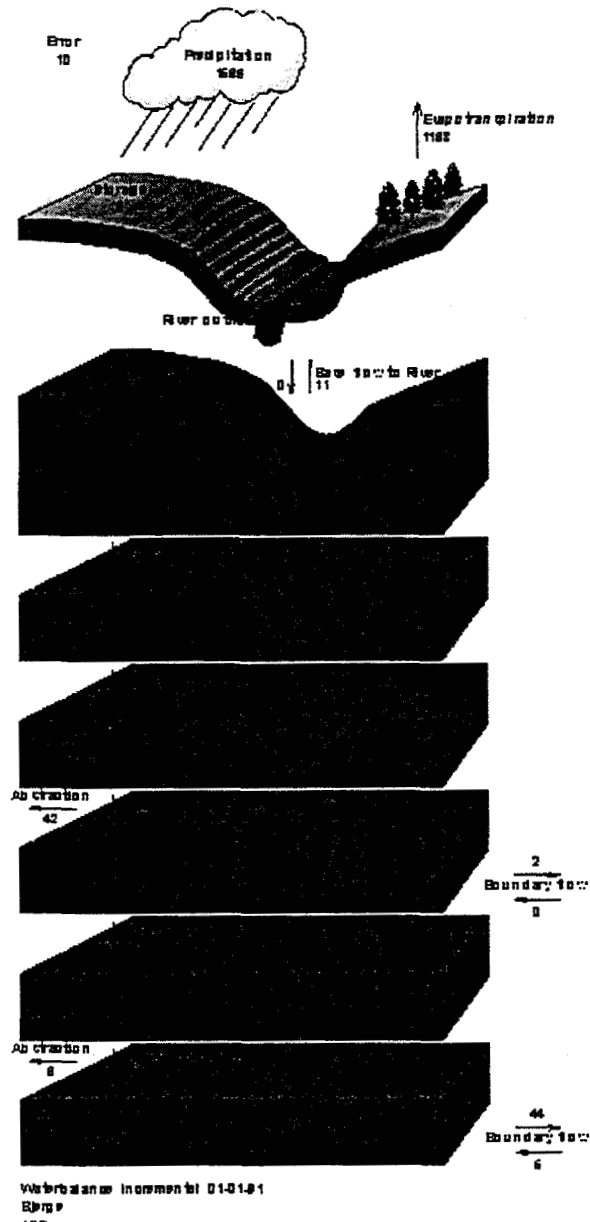


Figure 3 An example of the water balance chart. The chart shows the accumulated water balance the entire hydrological cycle: of six geological layers, of the ground surface, and of the atmosphere. The unsaturated zone is included in the most upper layer.

The Chart format is the file format for the water balance Chart Program. As for the T2 format all presented values are accumulated values from start date to end date. The water balance Chart Program only supports a few water balance types, see the config file for further information. An example of the water balance chart is given in Figure 3.





To run the water balance Chart Program, see Section 1.6.2.

### **1.3 Storage Requirement in the Flow Result File**

In order to achieve a correct water balance calculation a number of data types have to be stored in the Flow Result File. Table 2 shows the data types for each component that has to be stored.



Table 2

Required storage of data types. The user has to include all data types that are highlighted for the components included in the simulation. If e.g. the user wants to include snow melt (SM) in the simulation, it is a requirement to include data types 1 (precipitation) and 21 (snow storage) in the storing of results (menu F.2.a of the MIKE SHE menu system).

Data type	Component						Data type	Component					
	ET	SM	OC	RIV	UZ	SZ		ET	SM	OC	RIV	UZ	SZ
1							21						
2							22						
3							23						
4							24						
5							25						
6							26						
7							27						
8							28						
9							29						
10							30						
11							31						
12							32						
13							33						
14							34						
15							35						
16							36						
17							37						
18							38						
19							39						
20							40						

## 1.4 Configuration File

The configuration file specifies the contents of the defined water balance type. For given water balance types a number of groups are specified. Each group consists of one or more water balance items. Group items are added together (or subtracted) in the presentation. Component specification is required for all items. A predefined set of water balance types is present in the file \she540\bin\config.wbl. A complete overview of all water balance items is listed in Section 1.5. Groups can have arbitrary names, while item names has to be identical to the names listed in Section 1.5.

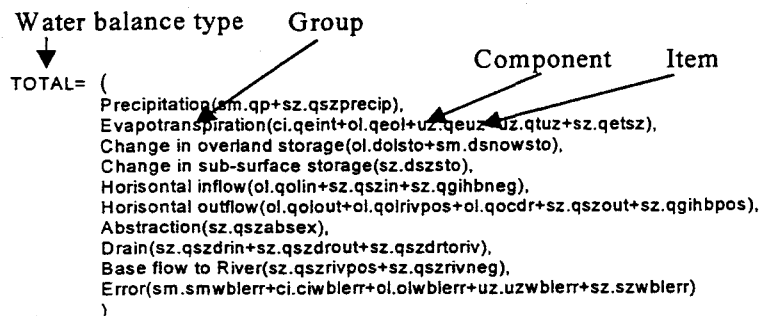


Figure 4 Example of water balance type specification. A group could be "Evapotranspiration", a component could be "ol", and an item could be "qeol".

## 1.5 Water Balance Items

The water balance extract program (mshe\_wbl\_ex.exe) calculates component water balance for the snowmelt component, the canopy interception component, the overland component, the unsaturated zone component, and the saturated zone component. Water balance calculations are performed at the maximum storing time step given in the water movement module (WM).

Recall that all units are given in millimetres of water per area.

The water balances for each component are written to so-called gross files. This section lists the water balance terms that are present in the gross files.



Snowmelt Terms (SM)		
<i>Item</i>	<i>Description</i>	<i>Sign</i>
qp	Precipitation	Positive upwards
qirrsprinklerin	Sprinkler Irrigation from source inside the sub-catchment	Positive out (always negative)
qirrsprinklerex	Sprinkler Irrigation from source outside the sub-catchment	Positive out (always negative)
qpad	Precipitation deducted snow and evaporation	Positive upwards
dsnowsto	Change in snow storage	Positive when increasing
qesnow	Evaporation from snow	Positive out (not implemented!!!!)
smwblerr	SM Water balance error	Positive when the component generates water

Canopy Interception Terms (CI)		
<i>Item</i>	<i>Description</i>	<i>Sign</i>
qpad	Precipitation deducted snow	Positive upwards
qpnet	Canopy through-fall	Positive upwards
qeint	Evaporation from intercepted water	Positive out
dintsto	Change in interception storage	Positive when increasing
ciwblerr	CI Water balance error	Positive when the component generates water



### Overland Terms (OL)

<i>Item</i>	<i>Description</i>	<i>Sign</i>
qpnet	Canopy through-fall	
qirsheetin	Sheet Irrigation from source inside the sub-catchment	Positive out (always negative)
qirsheetex	Sheet Irrigation from source outside the sub-catchment	Positive out (always negative)
qirrdripin	Drip Irrigation from source inside the sub-catchment	Positive out (always negative)
qirrdripex	Drip Irrigation from source outside the sub-catchment	Positive out (always negative)
qeol	Evaporation from ponded water	
qolin	OL potential flow into sub-catchment	
qolout	OL potential flow out of sub-catchment	
qolrivpos	Overland flow to river	
qocdr	Overland flow directly to river (from paved areas)	
qh	Infiltration from OL to UZ	
qolszpos	Upwards potential flow from SZ to OL	
qolszneg	Downwards potential flow from OL to SZ	
qsztofloodpos	SZ flow to flooded areas	
qsztofloodneg	flow to SZ from flooded areas	
qfloodtorivin	Exchange from overland flooded areas to river inside the sub-catchment	
qfloodtorivex	Exchange from overland flooded areas to river outside the sub-catchment	
dolsto	Change in overland storage	
olwblerr	OL water balance error – positive when the component generates water	



River Terms (RIV)		
<i>Item</i>	<i>Description</i>	<i>Sign</i>
qrivflowin	River flow into the sub-catchment [mm]	
qrivflowout	River flow out of the sub-catchment [mm]	
drivsto	Change in river storage	
qrivsinksource	External sink/sources to river	
qsztorivin	Base flow from SZ to River (Internal flow to river (Same sub-catch))	
qsztorivex	Base flow from SZ to River (External flow to river)	
qrivtoszin	Base flow from River to SZ (Internal flow to river)	
qrivtoszex	Base flow from River to SZ (External flow to river)	
qoltorivin	Overland flow to River (Internal flow to river)	
qoltorivex	Overland flow to River (External flow to river)	
qfloodtorivin	Flow from flooded areas to River (Internal flow to river)	
qfloodtorivex	Flow from flooded areas to River (External flow to river)	
qdraintorivin	Drain flow to River (Internal flow to river)	
qdraintorivex	Drain flow to River (External flow to river)	
qrivrrsheetin	Sink term to sheet irrigation inside the catchment	
qrivrrsheetex	Sink term to sheet irrigation outside the catchment	
qrivrrdripin	Sink term to drip irrigation inside the catchment	
qrivrrdripex	Sink term to drip irrigation outside the catchment	
qrivrrsprinklerin	Sink term to sprinkler irrigation inside the catchment	
qrivrrsprinklerex	Sink term to sprinkler irrigation outside the catchment	
rivwblerr	RIV water balance error – positive when the component generates water	

Unsaturated Zone Terms (UZ)		
<i>Item</i>	<i>Description</i>	<i>Sign</i>
qh	OL – UZ infiltration	
qrech	UZ - SZ recharge	
qtuz	Transpiration from root zone	
qeuz	Evaporation from soil	
qgwfeedbackuz	Feedback from linear reservoir (LR) to UZ	
dudzdef	Change in UZ deficit	
uzwblerr	UZ Water balance error	



Saturated Zone Terms (SZ)		
Item	Description	Sign
qrech	UZ – SZ recharge	
qsziprecip	Precipitation added directly to SZ	
qetsz	SZ evapotranspiration	
qszin	SZ potential flow into sub-catchment	
qsout	SZ potential flow out of sub-catchment	
qszzpos	Upwards SZ potential flow from layer i+1 to layer i	
qszzneg	Downwards SZ potential flow from layer i to layer i+1	
dszsto	Change in SZ storage	
szstocorr	UZ-SZ storage adjustment term – difference in unconfined storage capacity for UZ and SZ	
qolszpos	Upwards potential flow from SZ to OL	
qolszneg	Downwards potential flow from OL to SZ	
qsabsex	SZ abstraction	
qsdrin	SZ drainage into subcatchment	
qsdrout	SZ drainage out of subcatchment	
qsdrtoriviin	SZ drainage flow to rivers inside the subcatchment	
qsdrtorivex	SZ drainage flow to river outside the subcatchment	
qsdrivpos	SZ aquifer inflow to river	
qsdrivneg	River flow to SZ aquifer	
qgihbpos	Flow to General Internal Head Boundary cell	
qgihbneg	Flow from General Internal Head Boundary cell	
qirrshallowwell	Abstraction from shallow well to irrigation - positive out	
qirrremotewell	Abstraction from remote well to irrigation - positive out	
szwblerr	SZ water balance for layer i	
szwblerrtot	SZ water balance for sub-catchment - positive when model generates water	

## 1.6 Running the Water Balance Utility

The water balance utility consists of two console/DOS applications and a Windows application. There is no graphical user interface for the water balance execution implemented at the present stage. The steps of the execution of the water balance utility are outlined in the next sections.



### 1.6.1 Calculate the water balance

- Open a dos-prompt.
- Change directory to the MIKE SHE working directory (The one with the Dbase, Digfiles, Macro, etc. directories).
- Edit your macro file . . .
- Execute the extraction program by writing: "mshe\_wbl\_ex" The program will now prompt for the macro file name. Alternative write echo "<Macro file name> | mshe\_wbl\_ex" or simply "mshe\_wbl\_ex <runwbl.txt" where runwbl.txt is a text file containing the path and name of the macro file.
- Execution can be performed by utilising a batch program, see Figure 5.

```
echo Macro\91-93.wbl | mshe_wbl_ex > tmpfile
echo Macro\91-93.wbl | mshe_wbl_post > tmpfile
echo Macro\91.wbl | mshe_wbl_ex > tmpfile
echo Macro\91.wbl | mshe_wbl_post > tmpfile
echo Macro\92.wbl | mshe_wbl_ex > tmpfile
echo Macro\92.wbl | mshe_wbl_post > tmpfile
echo Macro\93.wbl | mshe_wbl_ex > tmpfile
echo Macro\93.wbl | mshe_wbl_post > tmpfile
echo Macro\91-93error.wbl | mshe_wbl_ex > tmpfile
echo Macro\91-93error.wbl | mshe_wbl_post > tmpfile
echo Macro\UZerror.wbl | mshe_wbl_ex > tmpfile
echo Macro\UZerror.wbl | mshe_wbl_post > tmpfile
del tmpfile
```

*Figure 5 An example of DOS commands assembled in a batch program (runwbl.bat) for automatic execution of a series of water balance investigations. In this sequence the water balance is calculated over a 3-year period (1991-1993) and for each of the individual years with chart illustration as purpose. The entire error contribution will be tabulated and the error distribution of the unsaturated zone will be mapped. The tmpfile contains the screen dump and is finally erased.*

- The execution of the post-processing program, mshe\_wbl\_post.exe, is analogue to the execution of the extraction program.
- Do ALWAYS check the SIGNALS directory for error files produced by the extraction or post-processing program.

### 1.6.2 Illustrate the water balance in a chart

To run the water balance Chart Program to create a water balance chart:

- Run the "mshe\_wbl\_ex" and "mshe\_wbl\_post" programs with output format = 4.





- Run the windows application "WblChart.exe" (\she540\bin\WblChart.exe).
- From the WblChart program: Open the wbl output file to show the chart.
- Notice: A shortcut of the WblChart program can be located on the desktop (target: C:\she540\bin\WblChart.exe, start in: C:\she540\bin). Double click the icon and open the wbl output file or simply "drag and drop" the wbl output file into the icon to show the chart.
- Notice: The chart may appear slightly different on screen compared to on print or print preview.



# **MIKE SHE PP – User Manual**

## **Data File Formats**

211





## CONTENTS

1	DATA FILE FORMATS.....	1
---	------------------------	---

2/2





## 1 DATA FILE FORMATS

### Time series

Four different ASCII file formats for time series data are used in MIKE SHE. Being ASCII files they can be easily edited in any text editor, spreadsheet etc.

For each data file type the first lines specifies which kind of data is in the data file. Each line in the header consists of 24 characters of text (which will be skipped when reading) followed by information;

- line 1 specifies the data file type, the data type (51 in all MIKE SHE input time series) and the version number (501 for MIKE SHE version 5.1);
- line 2 is a text line (only used to identify the data file for the user;
- line 3 defines the number of series (records) in the data file and a delete value which is the value used in a series with missing data;
- line 4 and 5 defines the start and end times respectively of the series as year, month, day, hour, minute.

The succeeding lines are different for the different data file types and will be described below.

### File type 2

File type 2 is the format of most input data for a simulation with the MIKE SHE WM module. It can consist of several series e.g. precipitation stations and is used to define precipitation data, potential evaporation data, root depth, leaf area index and temperature - one data file for each parameter - for a WM simulation. Also recorded time series of river discharge is stored as data file type 2. Time series retrieved from a Flow Result File with MSHE.OR and MSHE.WBL are also stored in data files as data file type 2.



```
FILETYPE DATATYPE Verno: 2 51 501
TEXTLINE           : Rootdepth (m)
NREC DELVAL        : 2 999.0
START DATE         : 1980 1 1 8 0
END DATE           : 1981 1 1 8 0
1960  1  1  8  0  1.0  3.0
1970  1  1  8  0  1.5 999.0
1980  1  1  8  0  1.5  3.0
1981  1  1  8  0 999.0  3.0
```

Figure 1 Example of time series for data file type 2.

The different input time series is defined in different units. The list given below defines this unit:

Time series: Unit:

precipitation mm/h  
evaporation mm/h  
root depth m  
leaf area index -  
temperature 0C

### File type 3

File type 3 is the format of recorded potential head data for which a table of the location of the records should be attached. Potential heads can be recorded as elevations (m.a.s) or as levels below a reference level and the locations should be given as an UTM-co-ordinate.

- line 6 of a data file of this type defines the UTM zone (specify a zero if unused) and the xy-units and level type:

xy-units: 1 - x and y units are kilometres  
          2 - x and y units are metres

z-type: 1 - level is elevation  
         2 - level is determined as levels below a reference level

z-unit is always metres

The succeeding (number of series) lines determines the location of the monitoring wells and should be given as:



inum, x-location, y-location, reference level, ilay, identification text as shown in the figure below.

```

FILETYPE DATATYPE VERNO: 3 51 501
TEXTLINE                : potential head observations.
NREC DELVAL              : 4 -999.
START DATE               : 1960 6 1 8 0
END DATE                 : 1967 6 1 8 0
UTM XYUNIT ZTYPE        : 34 1 2
1 11.10 7.90 125.63      4 'record id 1'
2 11.70 7.80 101.83      4 'record id 2'
3 11.40 7.10 109.26      4 'record id 3'
4 12.70 10.30 126.99     4 'record id 4'
1960 6 1 8 0 1.25 1.25 1.25 1.25
1961 6 1 8 0 -999. -999. -999. -999.
1963 6 1 8 0 -999. -999. -999. 1.25
1964 6 1 8 0 1.25 -999. -999. 1.25
1965 6 1 8 0 1.25 -999. -999. -999.
1966 6 1 8 0 -999. -999. -999. -999.
1967 6 1 8 0 1.25 -999. -999. -999.

```

Figure 2 Example of time series for data file type 3.

#### File type 4

File type 4 is the format of groundwater abstraction data used as flux boundary conditions for a MIKE SHE WM simulation. Groundwater abstractions are defined as a rate with the unit 1000 m<sup>3</sup>/year i.e. if you specify 20 in the time series it corresponds to the rate 20,000 m<sup>3</sup>/year.

- line 6 of a data file of this type defines the UTM zone (specify a zero if unused) and the xy-units:

xy-units: 1-x and y units are kilometres  
 2-x and y units are metres

The succeeding (number of series) lines determines the location of the abstraction wells or groups of abstraction wells. A possible way of describing a group of abstraction wells is to define a "line" (x1,y1) - (x2,y2) covering the wells. MIKE SHE will then distribute the specified abstraction equally between those grid squares it touches. The specifications of location should be given as:

inum, x-location, y-location, reference level, ilay, x1-location, y1-location, x2-location, y2-location, identification text as shown in the figure below. Note that ilay determines the computational layer in which the flux boundary conditions are "working".

215





```
FILETYPE DATATYPE VERN0: 4 51 501
TEXTLINE           : groundwater abstraction
NREC DELVAL        : 4 -999.
START DATE         : 1965 2 1 0 0
END DATE           : 1967 6 1 0 0
UTM XYUNIT         : 34 1
1 25.3 13.2 .000 2 21.30 13.20 21.30 13.20 'idstring 1'
2 25.3 13.2 .000 2 21.30 13.20 21.30 13.20 'idstring 2'
3 20.3 8.1 .000 2 16.30 8.10 16.30 8.10 'idstring 3'
4 22.3 10.3 .000 2 18.30 10.30 18.30 10.30 'idstring 4'
1965 2 1 0 0 .100 .0 .450 .0
1965 3 1 0 0 -999.000 .200 .0 .050
1965 4 1 0 0 .050 .0 .450 .0
.
.
1967 6 1 0 0 .050 .0 .450 .0
```

Figure 3 Example of time series for data file type 4.

### Matrix data

In the present version (5.1) of MIKE SHE there is only one format for matrix data but the data files are still ASCII formatted and can be inspected and modified with a normal text editor. Compared to earlier releases of SHE the heading lines of the matrix data are changed.

Matrix data values represent most often a mean value over a certain area - the area of the Grid Square. Since MIKE SHE can operate on sub-areas or small parts of a "file" it is necessary to define the dimensions **and** the origin of the data in the data file.

The heading line (5) specifies which kind of data is in the data file. Each line in the header consists of 24 characters of text (which will be skipped when reading) followed by information;

- line 1 specifies the data file type, the data type and the version number (501 for MIKE SHE version 5.1);

file type: 21 - the data in the data file is considered to be integers (grid codes) even though the data is written as reals

22 - the data in the data file is considered to be reals

data type: the data type is presently not used but the only allowable type is:

57: Any grid data (reals)



- line 2 specifies the dimensions and origin of the data;  
nx,ny: dimensions of the matrix  
dim: dimension of the grid squares [m]  
xorig,yorig: location of the origin in the  
co-ordinate system [km]
- line 3 defines a delete value which defines areas with missing values or outside the catchment, the UTM zone (specify a zero if unused) and the orientation of the matrix of your matrix data [degrees (positive anti-clockwise)]; You should be very careful with different orientations of your data files!!
- line 4 defines some statistical parameters for the data values in the data file - parameters which are output from the utility programs used to create a matrix data file;
- line 5 is a text line.

The succeeding lines are the data lines which has a format as shown in Figure 4 below. The values are written with a format of F7.x with 10 values in each row.

```
FILETYPE DATATYPE Verno:      21    100    501
NX NY DIM Xorig Yorig :      4    5    .500E+03    .00E+00    .00E+00
DELETE UTMZONE ORIENT :    -.100E-34      34      .000
MIN MAX MEAN ST.DEV   :    .20E+02    .70E+02    .45E+02    .418E+02
Text line to describe the data
5
-.1E-34-.1E-34-.1E-34-.1E-34
4
-.1E-34      70.      60.-.1E-34
3
-.1E-34      50.      40.-.1E-34
2
-.1E-34      30.      20.-.1E-34
1
-.1E-34-.1E-34-.1E-34-.1E-34
```

Figure 4 Example of a matrix data file.

### Digitized data

In the present version (5.1) of MIKE SHE there is still three different formats for digitized data and the data files are still ASCII formatted and can be inspected and modified with a normal text editor.



The heading lines (3) specifies which kind of data is in the data file. Each line in the header consists of 24 characters of text (which will be skipped when reading) followed by information;

- line 1 specifies the data file type, the data type and the version number (501 for MIKE SHE version 5.1);

file type: 31 - digitized contours  
32 - digitized polygons  
33 - old format for digitized river data

data type: (not used for digitized data)

- line 2 is a text line
- line 3 defines the UTM zone (specify a zero if unused) and the xy-units:

xy-units: 1-x and y units are kilometres  
2-x and y units are metres

```
FILETYPE DATATYPE Verno: 31 0 501
TEXTLINE          : Digitized contours
UTM XYUNIT        : 34 1
1 1.5 2.9 100.0 (dig. code,x,y,z)
2 1.6 2.8 100.0
2 1.8 2.7 100.0
2 1.9 2.6 100.0
3 2.0 2.3 100.0
1 4.6 1.3 90.0
2 4.0 1.2 90.0
3 3.9 1.3 90.0
```

Figure 5 Example of digitized data - data file type 31.



```
FILETYPE DATATYPE VERNO: 32  0  501
TEXTLINE           : Digitized polygons
UTM XYUNIT         : 34  1
1 1.5 2.0 1 (dig. code,x,y,polygon code)
2 1.5 3.3 1
2 1.8 3.2 1
2 1.9 2.6 1
3 1.5 2.0 1
```

Figure 6 Example of digitized data - data file type 32.

```
FILETYPE DATATYPE VERNO: 33  0  501
TEXTLINE           : Digitized river points (old format)
UTM XYUNIT         : 34  1
1.5 2.9 60.0      1 (x,y,z,dig.no)
1.6 2.8 999.999  2
1.8 2.7 999.999  3
1.9 2.6 70.0      4
2.0 2.4 71.0      5
2.1 2.5 72.0      6
```

Figure 7 Example of digitized data - data file type 33.

A special digitising format has been introduced in the graphical river editor for the description of a river cross-section. Presently, each digitized cross-section in the river editor must be stored in separate data files with the format shown in Figure 8. The format is as follows:

- line 1: Manning number for  $[m^{1/3}/s]$ ;
- line 2: leakage coefficient of the river lining  $[1/s]$ ;
- line 3++: x-z values of distance from the center of the river and the river bed level relative to the river bank elevation and both numbers in  $[m]$ .



```
20.000000
0.000000
-10.000000 0.000000
-8.000000 -0.200000
-6.000000 -2.000000
-2.000000 -2.000000
0.000000 -2.500000
2.000000 -2.000000
6.000000 -2.000000
8.000000 -0.200000
10.000000 0.000000
```

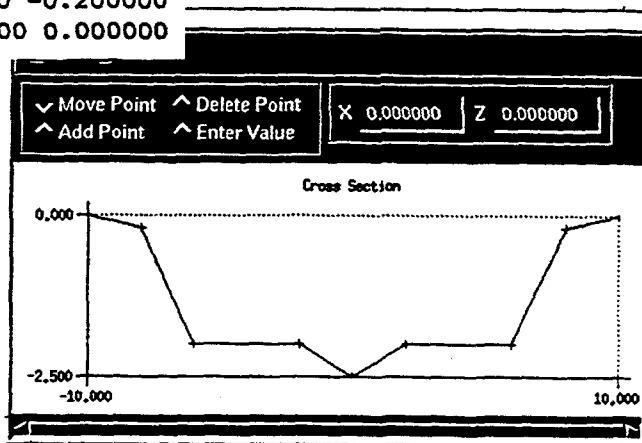


Figure 8 Example of digitized data for river cross-section and the corresponding cross-section.



# **MIKE SHE PP – User Manual**

## **Merget0 – Merging T0 Files**

221





## CONTENTS

1	MERGET0 - MERGING T0 FILES .....	1
---	----------------------------------	---

222







## 1 **MERGET0 - MERGING T0 FILES**

### **General Description**

To merge timeseries from a number of T0-files into one. It applies to T0-files of data type 2 - see the **Data File Format** section in this manual.

### **Methodology**

A number of time series files and record numbers are specified as input to the 'merge' utility. It reads the individual time series and writes them into one file taking into account varying storing intervals.

### **Input Data**

A number of T0-files.

### **Specifications**

A specification file containing the required input for the utility is given below

- (1) Output file name
- (2) Start time
- (3) End time
- (4) T0-file name and record number 1
- (5) T0-file name and record number 2
- .
- .
- ( ) T0-file name and record number n

For example

```
merge
1982 12 31 0 0
1983 12 31 0 0
TIME\prd.T0 1
TIME\qod.T0 1
TIME\prd.T0 3
```



The programme is executed by :

merget0 < {specification file name}

An output file is generated in the tmp subdirectory - in the above example, tmp\merge\_1.T0.



# **MIKE SHE PP – User Manual**

## **DIGTOSHE**

225





---

## CONTENTS

1	DIGTOSHE.....	1
---	---------------	---

226





## 1 DIGTOSHE

### General Description

For transformation of data xyz data to MIKE SHE format for digitized data file type 31, 32 and 33. For instance DIGTOSHE can transfer output from the DIGI program (see the **Digitising using the DIGI Program** section for a description of DIGI in this manual).

### Methodology

DIGTOSHE reads and ASCII formatted file with digitized data stored as:

x1 y1 z1 No1

.

.

xn yn zn Non

where x, y, z are real values and No is an integer. x and y are the coordinates of the digitized points and z may be a level (e.g. topographical level) or a code value (e.g. vegetation areas).

The data are processed dependent on the desired output file format as listed below (see the **Data File Format** section in this manual):

File type: Format:

31 code x y z

32 code x y z code 2

33 x y z No

code: identifies start and end points of a contour or a polygon (1=start, 2=point, 3=end);

zint: is the same as z but transformed to an integer value;

No: is only used for file type 33 (digitized river points) to identify each point

### Input data

An ASCII-formatted data file containing the digitized data.

227





## **Output data**

A data file containing digitized data in the MIKE SHE format.

## **Specifications**

- (1) Input data file;
- (2) Output data file;
- (3) Output type:  
1: file type 31;  
2: file type 32;  
3: file type 33.
- (4) UTM zone
- (5) x/y unit  
1: km  
2: m
- (6) Undefined value  
Only output type 3:  
z-value of river points without bank elevation



# **MIKE SHE PP – User Manual**

## **DIGTRANS**

229





---

## CONTENTS

1	DIGTRANS.....	1
---	---------------	---

230





## 1 DIGTRANS

### General Description

For transformation of digitized data from one co-ordinate system to another.

### Methodology

DIGTRANS reads a DIGI output data file and after a co-ordinate transformation writes the data to an output data file of the same format.

### Input Data

A DIGI output data file.

### Specifications

Specify the following items:

- (1) Input data file name;
- (2) Output data file name;
- (3) Input data format: xy or utm. If xy is specified the program assumes that the data are written as x,y in the input data file. If utm is specified the program assumes that the data are written as y,x in the input data file;
- (4) Output data format: xy or utm. If xy is specified the program written the data as x,y in the output data file. If utm is specified the program writes the data as y,x in the output data file;
- (5) The transformation of the x-coordinates: Xadd, Xscale; the new x-co-ordinates will be transformed to  $X_{new} = (X_{old} + X_{add}) * X_{scale}$ ;
- (6) The transformation of the y-co-ordinates: Yadd, Yscale; the new y-co-ordinates will be transformed to  $Y_{new} = (Y_{old} + Y_{add}) * Y_{scale}$ ;

231



- (7) The rotation (degrees, positive anti-clockwise) around the new origin.

### **Output Data**

DIGTRANS produces an output data file written in the same format as output data files from the DIGI program.



# **MIKE SHE PP – User Manual**

## **Running MIKE SHE Programs in Batch Mode**







---

## CONTENTS

1	RUNNING MIKE SHE PROGRAMS IN BATCH MODE.....	1
---	--	---

234





## 1 **RUNNING MIKE SHE PROGRAMS IN BATCH MODE**

For experienced users it may be faster to prepare specification files for the different tool with a text editor or make minor changes in existing specification files "behind" the menu system. However, it should be emphasized that the checks, warnings and error messages will not guide you through this exercise.

In order to help the experienced users this appendix contains a description of the specification files for the mostly applied tools and a guide how to execute the tool. The description is very short and if you have problems about the understanding of e.g. data types etc. it is recommended to apply the tool from the menu system. Especially the graphical presentation tool might give some problems because of the variety of different presentations this tool offers.

Application of specification files allows the user to submit a simulation and retrieve and/or plot results from the simulation without activating MIKE SHE. You can for instance prepare a specification file which runs a simulation, retrieves data for water balance and prepares the presentations.

The MIKE SHE WM and MIKE SHE AD requires no specification file. These programs are executed by entering:

```
echo <name.wm> | mshe.wm  
echo <name.ad> | mshe.ad
```

where name.wm is the prefix of the flow input file and name.ad is the prefix of the data files which gives the specifications to the advection-dispersion simulation (MUST be located on a directory with the name AD!).

The following sections describe the input file formats for some of the MIKE SHE programs. In general the easiest way to generate an input file is to use the <save> facility in MIKE SHE's user-interface and subsequently modify the saved "template"-file.

235





## MSHE.IR (input retrieval)

### UZ Data as a Profile

Line	Variable
1	name of the flow input file;
2	'0' (to indicate that it is UZ data);
3	output point (ix , iy);
4	data type to be retrieved according to the list given below: 0: soil properties; 1: initial water content;
5	name of the output data file.

### SZ Data as a Profile

Line	Variable
1	name of the flow input file;
2	'1' (to indicate that it is SZ data as profiles);
3	output point (ix , iy);
4	data type to be retrieved (only 0 is possible): 0: hydrogeological parameters;
5	name of the output data file.

### SZ Data as Matrix Data

Line	Variable
1	name of the flow input file;
2	'2' (to indicate that it is SZ data as maps);
3	layer to be retrieved;
4	data type to be retrieved according to the list given below; 0: boundary grid codes; 1: levels of the computational layers; 2: horizontal hydraulic conductivity; 3: vertical hydraulic conductivity; 4: specific yield; 5: specific storage coefficient; 6: initial potential head; 7: boundary x-fluxes; 8: boundary y-fluxes; 9: boundary x-gradients; 10: boundary y-gradients;
5	name of the output data file.



## Other Data as Matrix Data

Line	Variable
1	name of the flow input file;
2	'3' (to indicate that it is other data as maps);
3	data type to be retrieved according to the list given below;
	0: surface topography;
	1: Manning numbers for overland flow;
	2: detention storage for overland flow;
	3: drainage levels;
	4: drainage constants;
	5: grid codes for catchment boundary;
	6: grid codes for precipitation stations;
	7: grid codes for evaporation stations;
	8: grid codes for temperature stations;
	9: grid codes for vegetation types;
	10: grid codes for soil profile types;
	11: grid codes for paved areas;
	12: grid codes for bypass areas;
	13: grid codes for unsaturated zone calculation profiles;
4	name of the output data file.
	<i>haslev2.frf</i> <i>haslev2.frf</i>
	0                                      1
a)	20 20                      b) 20 20
	0                                      0
	<i>test2.ir</i> <i>test2.ir</i>
	<i>haslev2.frf</i> <i>haslev2.frf</i>
	2                                      3
c)	1                                      d) 0
	2 <i>test2.ir</i>
	<i>test2.ir</i>

Examples of specification files for the input retrieval program, MSHE.IR.

- a) UZ data
- b) SZ data as a profile
- c) SZ data as matrix data
- d) Other data as matrix data

The program can be executed by entering:

mshe.ir < {specification file name}



## T2OPR (operations on matrix data)

Line	Variable
1	constant 1 (addition);
2	constant 2 (multiplication);
3	constant 3 (multiplication);
4	operand: 1: addition; 2: multiplication; 3: division;
5	input data file 1;
6	input data file 2;
7	output data file;
8	grid code data file.

1
2
3
1
<i>datafile1</i>
<i>datafile2</i>
<i>outputfile</i>
<i>gridcodefile</i>

Example of a specification file for the matrix operation program, T2OPR.

The program can be executed by entering:

mshe.t2opr < {specification file name}

239





### T0OPR (operations on time series)

Line	Variable
1	constant 1 (addition);
2	constant 2 (multiplication);
3	constant 3 (multiplication);
4	operand: 1: addition; 2: multiplication; 3: division;
5	input data file 1;
6	record number in data file 1;
7	input data file 2;
8	record number in data file 2;
9	output data file;
10	start date for operation;
11	end date for operation.

```
1
2
3
1
datafile1
1
datafile2
1
outputfile
1990 1 1 0 0
1991 1 1 0 0
```

Example of a specification file for the operations on time series program, T0OPR.

The program can be executed by entering:

mshe.t0opr < {specification file name}



### MSHE.OL (overlay)

Line	Variable
1	origo and orientation of coordinate system (xbeg. , ybeg. , orientation);
2	number of elements and dimension (nx , ny , dimension [m]);
3	add boundary: 0: no; 1: yes;
4	initial code value;
5	input data file with digitized data;
6	{grid code data file};
7	output data file.

```
0 0 0
10 20 30
0
5
inputdatafile
gridcodefile
outputfile
```

Example of a specification file for the overlay program, MSHE.OL.

The program can be executed by entering:

mshe.ol < {specification file name}



### MSHE.BND (extraction of boundary conditions)

Line	Variable
1	flow result file (frf);
2	flow input file (fif);
3	extraction time step [hours];
4	extraction period [hours from start date for simulation with the fine model];
5	output data file.

*haslev3.frf*  
*test2.fif*  
*720*  
*0 61320*  
*outputfile*

Example of a specification file for the boundary extraction program, MSHE.BND.

The program can be executed by entering:

mshe.bnd < {specification file name}



## T2INTP (interpolation of two-dimensional data)

Line	Variable
1	origo and orientation of co-ordinate system (xbeg. , ybeg. , orientation);
2	number of elements and dimension (nx , ny , dimension [m]);
3	interpolation search radius [m];
4	data type according to the list given below: 60: Surface topography; 61: Geology layers; 62: Horizontal hydraulic conductivity; 63: Vertical hydraulic conductivity; 64: Unconfined storage coefficient; 65: Confined storage coefficient; 66: Drainage level; 57: Other T2 data;
5	data type text according to the list given above;
6	{grid code data file};
7	output data file;
8	number of input data files (max 6);
9	input data file 1;
10	input data file 2; etc. etc.

```
0 0 0
62 70 500
2000
60
'Surface topography'
```

```
MAPS/topotest.T2
1
DIGFILES/TOPOGRAPHY.DIG
```

Example of a specification file for the two-dimensional interpolation program, T2INTP.

The program can be executed by entering:

mshe.t2intp < {specification file name}

243



## MSHE.WBL (water balance calculations)

### General

Line	Variable
1	flow result file (frf);
2	water balance type according to the list given below: <ul style="list-style-type: none"><li>1: total water balance;</li><li>2: water balance error in the different components;</li><li>3: water balance for each layer in the saturated zone component [mm];</li><li>4: as 3) but values are calculated as <math>m^3</math>;</li><li>+100: water balances are calculated for a subcatchment;</li><li>+1000: water balances are written to the output data file in T0 format;</li><li>x-1: water balances are calculated as incremental values;</li></ul>
3	output time step [hour];
4	output period [hours from start of simulation];
5	output data file.

### Water balance for a subcatchment:

3	grid code file name;
4	output time step [hour];
5	output period [hours from start of simulation];
6	output data file.

### Water balance from the SZ component and output as T0 format:

3	output layer;
4	output time step [hour];
5	output period [hours from start of simulation];
6	output data file.



**Water balance from the SZ component for a subcatchment and output as T0 format:**

3 grid code file name;  
4 output layer;  
5 output time step [hour];  
6 output period [hours from start of simulation];  
7 output data file.

	<i>test2.frf</i>		<i>test2.frf</i>
	3		103
a)	240	b)	MAPS/subgrid.T2
	0 8640		240
	outputfile		0 8640
			outputfile

	<i>test2.frf</i>
	-1103
c)	MAPs/subgrid.T2
	1
	240
	0 8640
	outputfile

Example of specification files for the water balance program, MSHE.WBL.

- a) general
- b) SZ sub-catchment
- c) SZ sub-catchment, output in T0-format

The program can be executed by entering:

mshe.wbl < {specification file name}

245



## MSHE.GD (graphical presentation)

The graphical presentation tool contains both specifications for time series and matrix data presentation. If you mix the two you should always specify the time series data first.

The specifications for each image can be given in several "overlays" and each overlay should follow the rules for either time series overlays or matrix overlays.

### General

Line	Variable
1	flow result file (frf) - if any;
2	device number;
3	number of images in the horizontal and vertical direction (for automatic scaling);
4	size of device - horizontal and vertical (mm);
5	start date for simulation;
6	end date for simulation;
7	text strings for "bottom text": 1: header; 2: date; 3: user;
8	logical parameter for legends on the plots [y/n];
9	number of time series overlays specified and number of images specified;
10	number of matrix overlays specified and number of plots;
11	logical parameter for read of data from a simulation result file [y/n];

### Specifications for time series overlays:

t1	image number, data type (see Chapter 6), data file name (if any), x-coordinate in the grid or record number if you are presenting observed data, y-coordinate in the grid, layer number in the grid, item number (only for data type 16 and 18): 1: x-flow; 2: y-flow; 3: z-flow (only data type 16); 4: abstraction (only data type 16); 5: groundwater / river exchange flow (only data type 16); line type (see Chapter 6), special option parameter: 0: no special options are given and line t2 to t14 should not be specified;
----	---



- 1: special options are given and line t2 to t14 should be specified;
- t2 start date for this overlay;
- t3 end date for this overlay;
- t4 ymin, ymax, ydelta and number of decimals on y-axis numbers;
- t5 y-axis text;
- t6 multiplication and summation factors for time series data;
- t7 header text on the image;
- t8 text on the legend;
- t9 sizes (x-size and y-size [mm]) and location (x-origo and y-origo [mm]) of the image;
- t10 sizes of the image texts (x-units, y-units, y-axis text, header text, legend text [mm]);
- t11 logical parameters for frame, grid, axis, x-units, y-units, tic marks [t/f];
- t12 colour scale and number of colours (should only be different from 0 0 if data type is 24 (water content in the unsaturated zone));
- t13 colour numbers (should only be different from 0 if data type is 24 (water content in the unsaturated zone));
- t14 data intervals (should only be different from 0 if data type is 24 (water content in the unsaturated zone));

#### Specifications for matrix and cross-section overlays:

- g1 image number, data type (see Chapter 6), data file name (if any), layer number in the grid (or number of layers if cross-section overlay), item number (only for data type 16 and 18:
- 1: x-flow;
- 2: y-flow;
- 3: z-flow (only data type 16);
- 4: abstraction (only data type 16);
- 5: groundwater / river exchange flow (only data type 16);
- line type (see Chapter 6), special option parameter:
- 0: no special options are given and line t to t should not be specified;
- 1: special options are given and line t2 to t should be specified;
- g2 date for this overlay (only used for simulated data);
- g3 image area (x-start, x-end, y-start, y-end in the grid) or specification of a cross-section and number of bends in the cross-section;
- g3a if the number of bends in the cross-section (n) is larger than 1 give then n-1 lines with the end points of each section;
- g4 ymin, ymax, ydelta and number of decimals on y-axis numbers (only used for cross-sections);





g5 multiplication and summation factors for matrix data;  
g6 header text on the image;  
g7 text header for the legend;  
g8 sizes (x-size and y-size [mm]) and location (x-origo and y-origo [mm]) of the image;  
g9 sizes of the image texts (x-units, y-units, y-axis text, header text, legend text [mm]);  
g10 logical parameters for frame, grid, axis, river network, x-units, y-units, tic marks [t/f] on the image;  
g11 colour scale and number of colours;  
g12 legend text for each data interval (only used for presentation of "areas");  
g13 colour numbers;  
g14 data intervals;

```
'test2.tif'  
0  
200 200  
2 2  
1990 1 1 0 0 24  
1991 1 1 0 0  
" " "  
'y'  
2 1  
3 2  
'y'  
1 51 'TIME/prd.T0' 1 0 0 0 201 1  
1981 3 1 24 0 T  
1981 3 31 24 0  
0 1 0 1  
'mm/hour'  
1 0  
'Precipitation and potential evaporation'  
'precipitation'  
140.0 30.0000 25 15  
2.5 2.5 2.5 3 2.5  
F F T T T T  
0 0  
0  
0  
1 51 'TIME/epd.T0' 1 0 0 0 200 1  
1981 3 1 24 0 T  
1981 3 31 24 0  
0 1 0 1  
'mm/hour'  
1 0  
'Precipitation and potential evaporation'  
'evaporation'  
140.0 30.0000 25 15  
2.5 2.5 2.5 3 2.5  
F F T T T T  
0 0  
0
```



```
0
2 71 " 1 1 2 1
1990 1 1 0 0
1 31 1 35 1
0 0 0 2
1 0
'Distribution of the precipitation stations'
'Legend'
62.0000 70.0000 25 60.0000
2.5 2.5 2.5 3 2.5
F T T T T T F
0 8
1 2 3 4 5 6 7 8
'station 1' 'station 2' 'station 3' 'station 4' 'station 5' 'station 6' 'station 7' 'station 8'
1 2 3 4 5 6 7
2 90 'DIGFILES/CATCHMENT.DIG' 1 1 25 1
1990 1 1 0 0
1 31 1 35 1
0 0 0 2
1 0
'Distribution of the precipitation stations'
"
62.0000 70.0000 25 60.0000
2.5 2.5 2.5 3 2.5
F T T T T T F
0 0
0
"
0.0
3 61 " 1 1 100 1
1990 1 1 0 0
15 28 35 1 1
-25 100 0 0
1 0
'Section 1'
"
150 40.0000 25 150
2.5 2.5 2.5 3 2.5
F T T T T T F
0 1
3
"
0.0
```

Example of a specification file for the graphical presentation program MSHE.GD. The specification file contains specifications for two time series overlays in one image and two matrix overlays and one cross-section in two images.

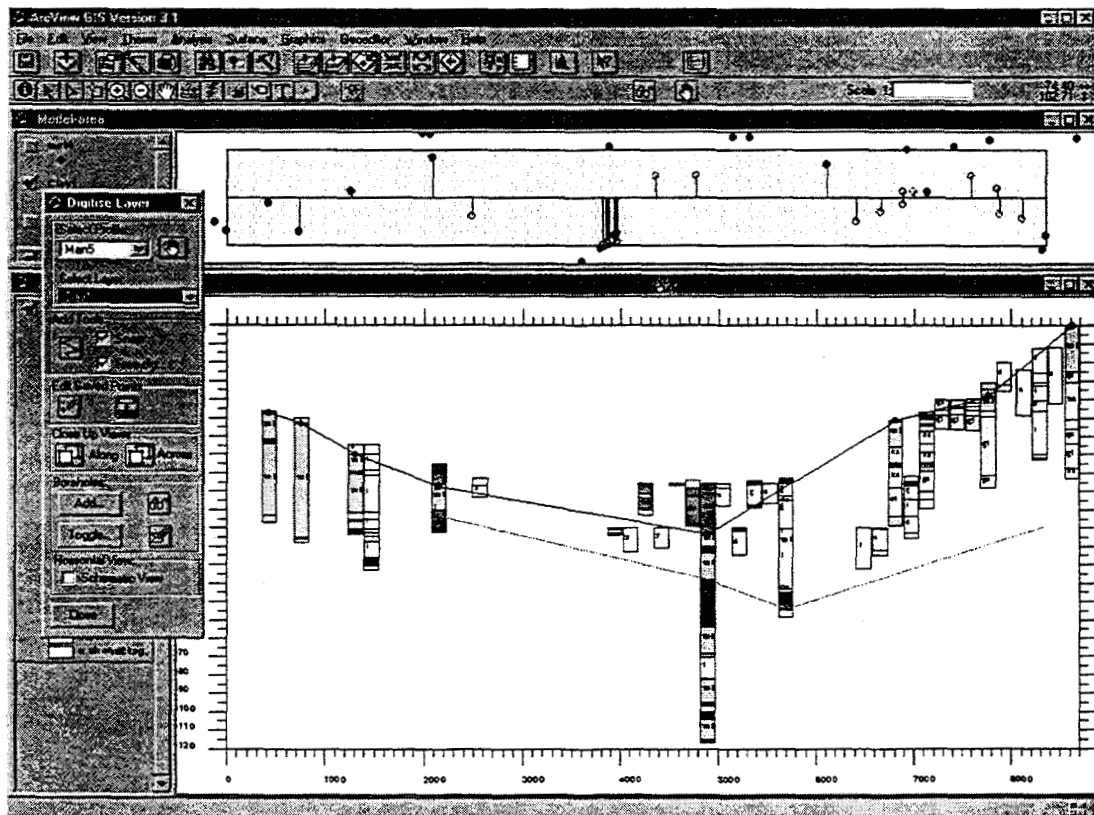
The program can be executed by entering:

mshe.gd < {specification file name}





# ArcView GeoEditor 1999 a



**A tool for Geological modelling and editing in GIS**

**Danish Hydraulic Institute**

**July 1999**

DHI Software 1999

ArcView GeoEditor 1999 a/Ed. 1.2

250





The GeoEditor is a product made by the Danish Hydraulic Institute (DHI) in co-operation with the Geological Survey of Denmark and Greenland (GEUS).

DHI is a private, non-profit research and consulting organisation providing a broad spectrum of services and technology in offshore, coastal, port, river, water resources, urban drainage and environmental engineering.

The warranty given by DHI is limited as specified in your Software License Agreement. The following should be noted: Because programs are inherently complex and may not be completely free of errors, you are advised to validate your work. When using the GeoEditor, you acknowledge that DHI has taken every care in the design of them. DHI shall not be responsible for any damages arising out of the use and application of the GeoEditor and you shall satisfy yourself that the GeoEditor provide satisfactory solutions by testing out sufficient examples.

The Danish Hydraulic Institute could be contacted by:

Danish Hydraulic Institute  
Agern Alle 5  
DK- 2970 Hørsholm  
Denmark

Phone: (+45) 45 17 91 00  
Fax: (+45) 45 76 25 67  
E-mail: [software@dhi.dk](mailto:software@dhi.dk)  
Home page: [www.dhi.dk](http://www.dhi.dk)





## CONTENTS

<b>1</b>	<b>INTRODUCTION .....</b>	<b>1</b>
<b>2</b>	<b>RELEASE NOTES .....</b>	<b>3</b>
<b>3</b>	<b>GETTING STARTED.....</b>	<b>5</b>
3.1	Requirements .....	5
3.2	Installation .....	5
3.3	How to Start GeoEditor.....	6
3.4	Views.....	6
3.5	Menus.....	7
<b>4</b>	<b>STARTING A NEW PROJECT .....</b>	<b>9</b>
4.1	Loading the GeoEditor .....	9
4.2	GeoEditor Menu .....	9
4.3	Defining the GeoEditor Folder.....	10
<b>5</b>	<b>INITIALISING THE GEOLOGICAL SET-UP.....</b>	<b>13</b>
5.1	Opening the New Project.....	13
5.2	Load Data Base Files .....	13
5.2.1	Recommended procedure .....	14
5.2.2	Content of the select data and model area dialog box .....	14
5.3	Initial Borehole Selection .....	15
5.3.1	Recommended procedure .....	15
5.4	Extracting the Initial Selection .....	15
5.5	Borehole Selection .....	16
5.5.1	Recommended procedure .....	17
5.5.2	Content of the borehole selection dialog box.....	18
<b>6</b>	<b>DEFINE GEOLOGICAL APPROACH.....</b>	<b>21</b>
<b>7</b>	<b>VERTICAL PROFILES.....</b>	<b>23</b>
7.1	Recommended Main Procedure.....	24
7.2	Define Geological Profiles .....	24
7.2.1	Recommended procedure .....	25
7.2.2	Content of the profile definition dialog box.....	26
7.3	Interpret Geology.....	27
7.3.1	Recommended procedure .....	28
7.4	Define Layers .....	29
7.4.1	Recommended procedure .....	30
7.4.2	Content of the define layers dialog box .....	30
7.5	Digitise Layers.....	31
7.5.1	Content of the digitise layers dialog box .....	32





<b>8</b>	<b>DEPTH INTERVAL.....</b>	<b>35</b>
8.1	Define Lithological Classes.....	36
8.1.1	Recommended procedure .....	37
8.1.2	Content of the define lithological classes dialog box.....	37
8.1.3	Not classified.....	38
8.1.4	Classified .....	38
8.1.5	Content of the classification of lithology dialog box.....	39
8.2	Lithology by Interval .....	40
8.2.1	Recommended procedure .....	40
8.2.2	Content of the lithology by interval dialog box.....	41
8.3	Specifying the Zonations.....	41
8.3.1	Recommended procedure .....	42
8.3.2	Content of the define zonations dialog box .....	42
8.3.3	Automatic generation of layer extension .....	45
8.3.4	Manually generation of layer extension.....	45
8.3.5	Edit zonation .....	45
<b>9</b>	<b>VALIDATION AND EXPORT.....</b>	<b>47</b>
9.1	Check Interpreted Points .....	47
9.1.1	Recommended procedure .....	47
9.1.2	Content of the check interpreted points dialog box .....	48
9.2	Surface Grid Set Up.....	48
9.2.1	Recommended procedure .....	49
9.2.2	Content of the grid set up dialog box .....	49
9.3	Convert to Surface.....	50
9.3.1	Recommended procedure .....	50
9.3.2	Content of the convert to surface dialog box.....	50
9.4	Check Surfaces.....	51
9.4.1	Recommended procedure .....	52
9.4.2	Content of the check surfaces dialog box .....	52
9.5	Adjust Surfaces.....	52
9.5.1	Recommended procedure .....	52
9.5.2	Content of the adjust surface dialog box.....	53
9.6	Replace delete Values .....	53
9.6.1	Recommended procedure .....	54
9.7	Export to XYZ File.....	54
9.7.1	Recommended procedure .....	54
9.7.2	Content of the export to XYZ file dialog box.....	55
<b>10</b>	<b>ADD DATA TO THE PROJECT .....</b>	<b>57</b>
10.1	Add GIS Data.....	57
10.2	Add Single Borehole .....	57
10.2.1	Recommended procedure .....	58
10.2.2	Content of add new borehole dialog box.....	59
10.3	Add Borehole Database.....	60
10.3.1	Recommended procedure .....	61
10.3.2	Content of the add data dialog box .....	61



10.3.3	Content of the select data dialog box .....	61
10.4	Borehole Selection .....	62
10.5	Add Geophysical Data.....	62
<b>11</b>	<b>BOREHOLE INFORMATION AND EDITING TOOL.....</b>	<b>63</b>
11.1	Recommended Procedure.....	63
11.2	Geology .....	64
11.2.1	Contents of the display borehole dialog box.....	65
<b>12</b>	<b>SETTINGS .....</b>	<b>67</b>
12.1	Axis Settings.....	67
12.2	Show Data Settings.....	68
12.3	GeoEditor Settings .....	70
12.4	Misc. Settings .....	71
12.5	Resistivity Filter .....	72
12.6	Lithology Colours.....	73
<b>13</b>	<b>TOOLS .....</b>	<b>75</b>
13.1	Tools for the Horizontal View.....	75
13.2	Tools for the Vertical View.....	76
13.3	Extract Data Tools.....	76
13.4	Print Profiles .....	78
13.4.1	Recommended procedure .....	78
13.4.2	The content of the export profile dialog box.....	78
<b>14</b>	<b>FILE CONVERSION .....</b>	<b>81</b>
14.1	Recommended Procedure.....	81
14.2	Content of the Convert Table Dialog Box.....	82
14.3	Content of the Define Fields Dialog Box.....	83
14.4	Content of the Define Screen Data Dialog Box .....	84
<b>15</b>	<b>TUTORIAL AND HELP SYSTEM .....</b>	<b>87</b>
15.1	Online Help.....	87
15.2	Tutorial Movies.....	87
15.2.1	Recommended procedure .....	87

254





# The GeoEditor

Instruction for the GeoEditor 1999 a  
Danish Hydraulic Institute, July 1999

## 1 INTRODUCTION

The GeoEditor is a graphical tool providing facilities to develop and test geological models based on borehole data and geophysical data. The GeoEditor provides a close link between basic geological and geophysical data, conceptual interpretation and model representation. The GeoEditor is based on an inherent methodology, which leads the user through a systematic definition of a geological model. Based on experience of geologists, hydrogeologists and modellers two alternative approaches based on specifying either overall geological structures or zonations of characteristic aquifer properties have been implemented.

Both of the approaches are divided in a three-phase approach where selection of data is followed by geological interpretation. The overall geological structure may be specified in terms of layers and lenses by stepwise sweeping through a number of predefined geological profiles or alternatively by specifying zones of characteristic aquifer types sequentially for a number of depth intervals from the ground surface to the deep aquifers. In the third step the discrete values are interpolated into a 3-dimensional geological model. The resulting geological model is, subsequently, evaluated. In case of rejection the geological model can be modified until acceptance. When the geological model is accepted it is transferred to input files of the groundwater model.

255





## 2 RELEASE NOTES

The purpose of the present release (GeoEditor 1999 a) was to correct minor errors and improve on the menu system rather than to implement a wide range of new features. The GeoEditor has been tested thoroughly by the Geological Survey of Denmark and Greenland and their feedback has been very valuable to implement in the GeoEditor. The outcome has been a considerably more stable and user-friendly program.

A couple of new features have been introduced:

- New validation options (check interpreted points and adjust surface to reference surface).
- Snap layer to surface theme.
- Snap to borehole option in the manual profile definition.
- Update all profiles when changing the view settings.
- Export of layer information from a point or a grid theme to an XYZ ASCII file.
- A Borehole Edit option. Enables editing of the lithology, screen position, and water level position for a selected borehole.
- A choice between Danish and English language mode of the field names in the 'Boreholes' theme (administrative data) and in the 'lit\_model.dbf' table (lithological data).
- A File Conversion extension. This will allow the user to import borehole database files and convert them into the PC-Zeus format supported by the GeoEditor.
- Introduction of a help and tutorial system.

256





### 3 GETTING STARTED

This user guide assumes that the user is familiar with the ESRI product ArcView® 3.1 or later. The user must be familiar with the extension Spatial Analyst 1.1 as well. Basic ArcView functions described in one of the ESRI manuals are not explained in the user guide, but is assumed to be known by the user.

#### 3.1 Requirements

The GeoEditor requires ArcView® 3.1 or later and the extension Spatial Analyst 1.1 or later. The recommended requirements to run the GeoEditor on a PC is a Pentium 166 MHz (or more) with at least 48 MB RAM and a minimum of 25 MB of free hard disk space.

#### 3.2 Installation

Double clicking on the setup.exe file, or the GeoEditor.exe file, on the enclosed CD-ROM does the installation of the extension. The set-up program will install the necessary files in the user-specified location. The GeoEditor is using a system variable called Shegis (pointing at a folder called Shegis), if this is already set the user should use the same Shegis directory for the installation of the GeoEditor.

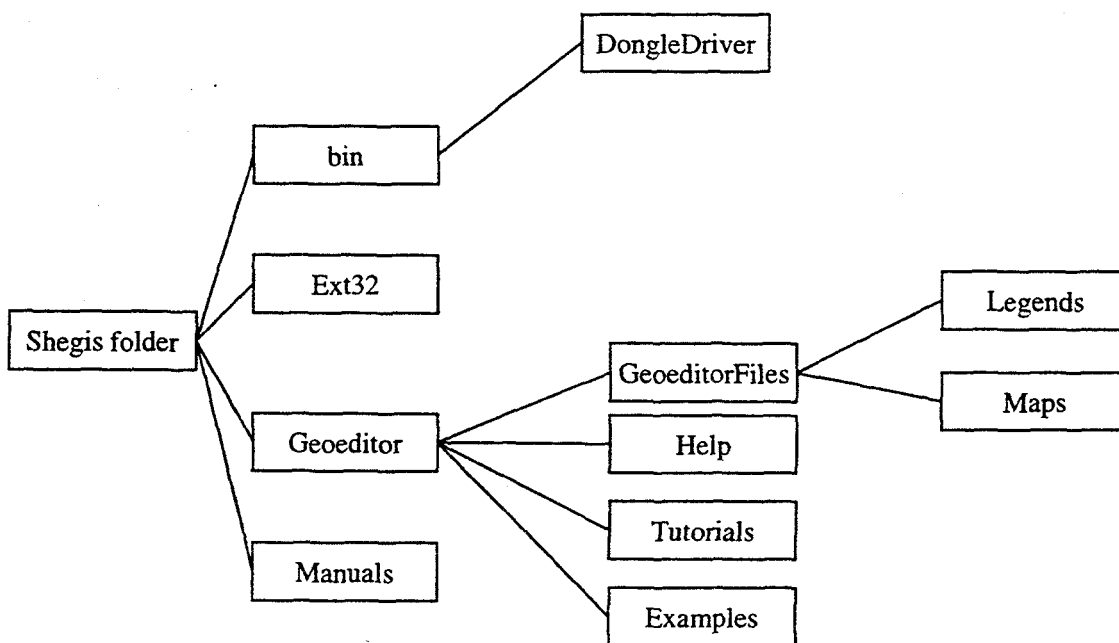


Figure 1 Folder Structure.





The Shegis folder is used commonly for all MIKE SHE related GIS products.

First the set-up will install the required files in order to make the Dongle Check; these files are saved in the bin folder. During the installation process the user is asked to locate the DHI Licence file, which is required in order to run the full version, this file is then saved in the bin directory.

The actual extension file DHIGeoeditor.avx, and other required avx files, are saved in the Ext32 folder. In order to use the GeoEditor the user has to move these files to the Ext32 directory (On windows based machines this is usually c:\Esri\Av\_gis30\Arcview\Ext32.), or in the userext folder, if this is created.

Two more folders called Manuals and GeoEditor are created, the Manuals folder contains the manuals, while the GeoEditor folder contains all the required files to use the GeoEditor, see Appendix 1 for a more detailed description of the files.

After finishing the set-up the user will have to restart the computer before using the GeoEditor.

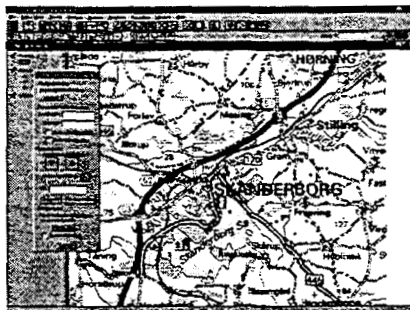
### **3.3 How to Start GeoEditor**

To invoke the GeoEditor the user must load it as an extension associated with ArcView. The extension menu is located under the file menu for the project window. Select the GeoEditor extension. During the load procedure the extensions Spatial Analyst and Dialog Designer will be automatically loaded, if they are not already loaded. If one of the extensions is not available the GeoEditor will fail to load. Also the system will check if the user has the required dongle and licence file in order to run the full version, if the dongle check fails the GeoEditor is opened in demo mode. In demo mode the user is only allowed to import a maximum of 10 boreholes.

### **3.4 Views**

The GeoEditor will create some specific views. These views are essential for using the GeoEditor. A GeoEditor process can not be performed in a user defined view. However, the user might add features to these views.

A short description of each of the views:

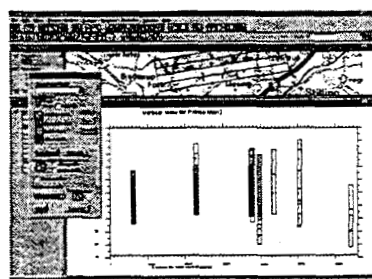


The **Model-area** view is the working-view for the GeoEditor. From this view the user is able to select a subset of boreholes, toggle between the profiling mode and define profiles and utilise most functionalities of the GeoEditor.

The user should not delete this view as all the information connected to the project goes with this view.

If the user chooses to define the geological model using the "Depth interval mode" the working-view will be a copy of the "Model-area" view, but renamed to **Horizontal-slice**.

When defining vertical profile this will appear in a special view, called "2D-Lithology view for" + the profile name. This view will by default be placed at the bottom of the screen occupying 80% of the screen. The remaining 20% will be occupied by the "Model-area" view, zoomed in to the profile.



From this view the user will be able to define the layers in the geological model, as discrete points.

Views of profiles can be removed manually from the project window.

### 3.5 Menus

All the menu functionality associated with the GeoEditor is in the GeoEditor menu. This menu has been structured to keep the user to the three-phase line of approach. The menus will be disabled or enabled according to the present situation, meaning that the user will not be allowed to perform an action that is not allowed in the situation.

259



Important Note: If the user is using the program EXCEED, it is recommended not to use the directory c:\exceed as the working directory in ArcView, because this could give some problems running the extension. The working directory is changed under the File menu, in the View.



## 4 STARTING A NEW PROJECT

### 4.1 Loading the GeoEditor

To use the GeoEditor the user must load it as an extension. When starting a new project the user will have to start the project using the GeoEditor menu in the project window. If the user makes a new window and tries to use the functions from here all the menu functions associated with the GeoEditor will be disabled.

### 4.2 GeoEditor Menu

When loading the GeoEditor extension a new menu, the GeoEditor menu, is added to the project window.

The GeoEditor menu contains 3 items, see Figure 2.

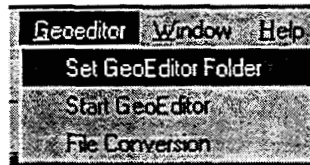


Figure 2 GeoEditor Menu.

**Set GeoEditor Folder:** Define the GeoEditor project folder, containing all the files created during a project. The user will have to define a project folder in order to proceed.

**Start GeoEditor:** Is enabled when a project folder is defined. This option starts the actual project by opening the **Model-area** view.

**File Conversion:** The file conversion option enables the user to convert a borehole database format to the PC-Zeus format (see Appendix 2) supported by the GeoEditor.

The tutorial and help menu connected to the GeoEditor is added to the window under the Help menu.

**Tutorial:** Enables the user to run some tutorial movies showing the basic facilities of the GeoEditor.

**Help:** Opens the on-line help system for the GeoEditor.



**About GeoEditor:** Opens the About GeoEditor dialog box, enabling the user to email questions or problems to the DHI software support centre and to open the DHI Internet site.

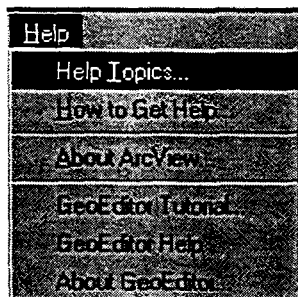


Figure 3 The Menus under the Help Menu.

### 4.3 Defining the GeoEditor Folder

Before the user is allowed to open a new project, a GeoEditor folder should be specified.

In the project view select the **Set GeoEditor folder** under the GeoEditor menu. Then the user will be presented to a dialog box showing the current project directory and a field for a new directory. A new project directory is defined by using the browse button or typing the path to the directory.

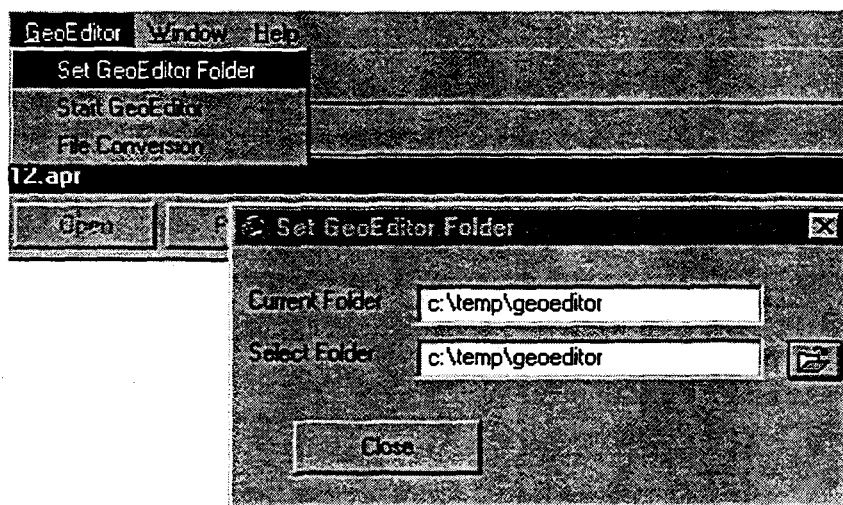


Figure 4 Define Project Directory.

In the project directory two sub-directories, Layers and Tmp will be made. All the files created when working with the GeoEditor are saved in the project directory.



When clicking **OK** the user will be prompted if the project file, the \*.apr file, should be saved in the newly defined project directory. If this is accepted the user is prompted to type a project name.

When a project directory is defined the **Start GeoEditor** option in the GeoEditor menu will be enabled.





## 5 INITIALISING THE GEOLOGICAL SET-UP

Initialising the geological set-up contains loading the source files, defining the initial selection of boreholes, and selecting a subset of the selection.

### 5.1 Opening the New Project

Selecting the **Start GeoEditor** menu under the GeoEditor menu in the project window starts the new project.

This action will open the **Model-area** view, see Figure 5. In this view the user should select the geological information source files and make the initial selection of boreholes.

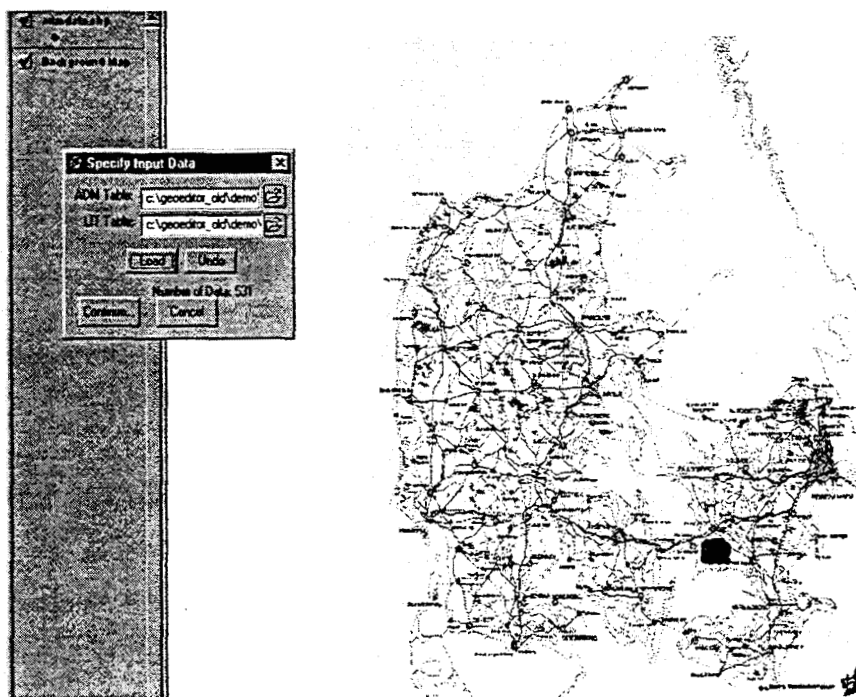


Figure 5 Model-area View.

### 5.2 Load Data Base Files

When opening a new project the first view will be the **Model-area** view, containing a geographic base-map and a dialog box. The purpose of this view is to load the source files.

By default the geographical base-map is a map of Denmark in 1:2.000.000. The base-map is loaded from the  
\\GeoEditor\\GeoeditorFiles\\Maps\\background.bmp, so to change the





default base-map a new bitmap file (bmp) should replace the default file.

### 5.2.1 Recommended procedure

- Select the **LIT** and the **ADM** files (the source files), by using the browse button or typing the paths in the dialog box.
- Click **Load** to display the boreholes from the source file on the map.
- Click **undo** to undo the borehole selection.
- Click **Cancel** to abort the GeoEditor project and return to the project view.
- Click **OK** to continue.

It is recommended to keep the number of selected boreholes below 1500. If the number of selected boreholes is too large the following processes will be very slow.

### 5.2.2 Content of the select data and model area dialog box

Data from the PZ-ZEUS database are stored in two separate files, **ADM**, and **LIT** file, see Appendix 2. The **LIT** file containing the lithological data while the **ADM** file keeps all the administrative information such as borehole position, owner, ID, screen, etc.

**Lit Table:** The user should define the lithological part of the database (\*.dbf file), by typing the file path or by selecting the browse button.

**Adm Table:** The user should define the administrative part of the database (\*.dbf file), by typing the file path or by selecting the browse button.

When the user presses the **Load** button, the two dBase files will be loaded, and a new theme will appear in the view. By default this theme will be visible, but the user can turn the theme off, by clicking in the Table of Contents (TOC).



### 5.3 Initial Borehole Selection

When the user press the **OK** button the extract selection dialog box will appear. The user should then select the initial borehole selection.

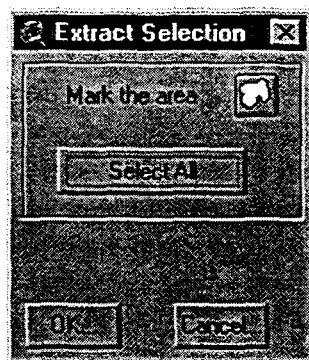


Figure 6 Extract Selection Dialog.

#### 5.3.1 Recommended procedure

- Select all the boreholes by pressing the **Select All** button.
- Select boreholes within a specified area by using the **Mark the area** tool.
- Press the **OK** button to extract the selected boreholes and continue.
- Press the **Cancel** button to abort the selection.

**Mark the area:** In order to select the model area the user should press the tool-button, and draw a polygon surrounding the approximate model area. Click on the view to draw the polygon. End the polygon drawing by double clicking. When the polygon is constructed the data inside it are automatically selected.

The number of boreholes in the source file and the number of boreholes in the current selection are displayed in the dialog box.

**OK:** End the first data selecting. Closes the view.

When the user clicks on the **OK** the selected data are extracted and presented as a new theme, the **Boreholes** theme.

### 5.4 Extracting the Initial Selection

When the user has made the initial boreholes selection the selected data are extracted from the original databases. The data from the **LIT** table are extracted and saved in an ArcView® table named **lit\_model.dbf**. The data from the **ADM** table are extracted to the



theme table associated with the active theme in the **Model-area** view, the **Boreholes** theme.

Depending on the number of selected data the extraction could take some time.

The user will be presented for the following dialog box during the process.

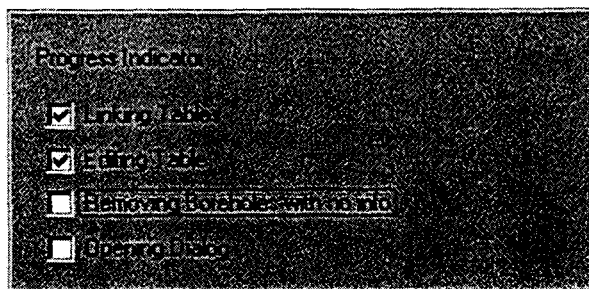


Figure 7 Progress Indicator.

**"Linking Tables"**: The extracted tables are linked.

**"Editing Table"**: A new field is added to the Boreholes theme. The Datastatus field used to describe the status of the borehole, the status could be: Data, User-defined or Added data. These descriptions will be further explained later.

**"Removing boreholes with no info"**: Removing the boreholes containing no geological information.

**"Opening Dialog"**: Opening the "Borehole Selection" dialog box.

Then the user is presented to the **Model-area** view, containing one theme, the Boreholes theme, representing the extracted data, and an open dialog box.

## 5.5 Borehole Selection

The borehole selection can be entered when opening a new project or if the user wants to redefine the borehole selection, see Figure 8.

The purpose is to select the boreholes that fulfil the criteria established by the user, e.g. if the user only wants to work with boreholes that contain limestone, are more than 10 meters deep and are constructed before March 1 1990.

The user may select the boreholes from different criteria. The selection only works on the extracted data, meaning that the user is selecting the subset of the extracted data that fulfil the established criteria.

When opening a new project, by default the boreholes containing no geological data will not be selected. However if the user presses the



**Select All** button they will be selected, although they do not contain any geological information.

At the bottom of the selection dialog box the total number of data and the amount selected are prompted. When the dialog box appears the number of selected data could be less than the total amount. This will occur in cases when data are missing in the lithological database or in case of boreholes containing blanks in the lithology field.

### **5.5.1 Recommended procedure**

- Click on a criterion, define the selection and press the **Select** button, or the **Remove** button. Several lines in the list could be selected by holding down the **SHIFT** button.
- If the desired selection criterion contains several subcriteria select them one by one. This makes the selection more fast, and it is easier to see the effect of the subcriterion.
- Use the **Select All** button to clear the current selection by selecting all the boreholes.

267



### 5.5.2 Content of the borehole selection dialog box

Figure 8 The Borehole Selection Dialog.

**Mark the area:** In order to select the model area the user should press the tool-button, and draw a polygon surrounding the approximate model area. Clicking on the view draws the polygon, end the polygon by double clicking. All the boreholes within the defined polygon will then be selected.

**Area Code:** Select by municipal code.

**Lith. Symbol:** Select by geological symbol, all the boreholes containing the selected geological symbol will be selected.

**Borehole ID:** Select by borehole ID.



**Type:** Select by borehole type e.g. abstraction or observation.

**Date:** Select by date. The user has to specify a before and an after date. The specification could be made from the list or by typing the dates in the two fields.

**Depth:** Select by depth. The user has to specify "Greater than" or "Less than", and then type a level (in the used data unit)

**Select:** Selection of data from the extracted data, which comply with the selection criteria.

**Remove:** Removes the data from the extracted data, which comply with the selection criteria.

**Select All:** Clears the actual selection, and selects all extracted data in the view.

**Retrieve:** When selecting the **Retrieve** button the initially selected boreholes will reappear. This can be useful, e.g. if the user first wants to do a geological interpretation only including the boreholes with limestone, and second wants to include other boreholes in the interpretation.

**OK:** To finish the selection press **OK**. When finishing all deselected data will be replaced from the **Boreholes** theme.  
Before entering the next part the user is asked to decide which kind of geological interpretation to employ.





## 6 DEFINE GEOLOGICAL APPROACH

When the initial definition of the geological set-up is ended the user is asked to decide which kind of geological interpretation to employ, see Figure 9.

**Vertical profiles:** Develop a geological model, consisting of geological layers interpolated from discrete points. Define horizontal profiles and digitise discrete points representing a selected lithology for each profile. Interpolate geological layers from the digitised points and export the layers to an external format.

**Depth intervals:** Develop surface maps representing the dominant lithology in specified layers. Classify the lithology, specify depth intervals for subtracting the dominant lithology, mark areas or define by Thiessen polygons and export to external format.

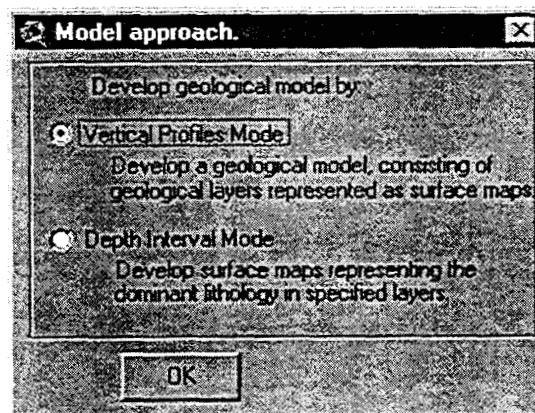


Figure 9 The Model Approach Dialog.

With respect to vertical profiles Chapter details 7 the geological set-up. With respect to depth intervals Chapter details 8 the development of surface maps. The validation and export of the interpretation is common to both approaches and is described in Chapter 9.

270







## 7 VERTICAL PROFILES

Define profiles and interpret the geology in each of the profiles, by defining the extent of each layer as discrete points. Interpolate the discrete points into surface maps representing the spatial extent of the layer, validate and export the map to an external groundwater model system.

The Vertical profiles mode is entered from the **Model approach** dialog box, see Chapter 6, or via the settings menu during a **Depth Interval** session, see Chapter 8.

When entering the Vertical profiles mode, see Figure 10, the main window will be the **model-area** view, initially only containing the **Borehole** theme.

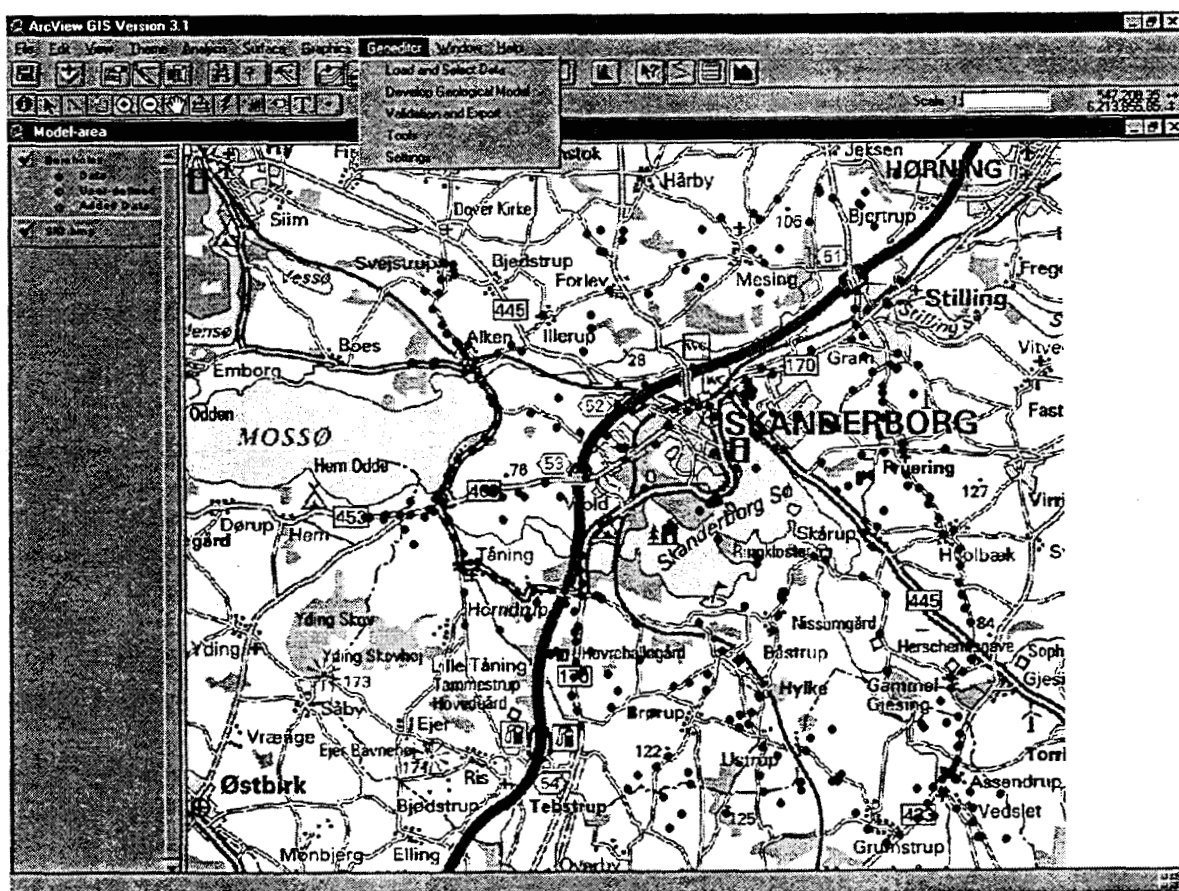


Figure 10 The Model-area View.

The menus associated with the interpretation are accessible through the GeoEditor menu, see Figure 10.



## 7.1 Recommended Main Procedure

### Load and Select Data (see Chapter 10)

- Import any external data first; topography, geophysical data, already interpreted layers or maps containing information about the area.

### Develop Geological Model (see Chapter 7)

- Define profiles.
- Edit the profiles, change the ID or delete the unwanted profiles.
- Make the vertical views, showing the boreholes along the selected profiles.
- Define the layers.
- Sweep through each of the profiles interpreting the geology.

### Validation and export (see Chapter 9)

- Converting the discrete points, representing the layers, to spatial maps (grids).
- Validate the maps; check for overlapping layers, import the maps to the vertical profiles to check the interpolation.
- Adjust surface to reference surface.
- Export the maps to an external format, XYZ ASCII file.

## 7.2 Define Geological Profiles

When using the Vertical Profiles approach the basis for the geological model is the definition of the geological profiles. In the GeoEditor the profiles are defined as a profile line with a user defined bandwidth. All the boreholes within the distance of a bandwidth from the profile line are projected into the profile line. This also applies for TEM data added with the profile. This means that the user should take caution when specifying the bandwidth. If it is too large, boreholes with different preconditions could be added to the same profile making the geological interpretation more difficult. If it is too small, the number of boreholes contributing to the profile eventually may be limited.

The geological profiles are defined in the **Profile** dialog box, accessible from the Develop Geological Model menu through **Define Profiles**.

When the dialog box activates, see Figure 11, the **Profiles** theme is created. This theme contains polygons representing the profiles, and the information associated with each of the profiles. The theme will not be created if the theme already exist when the dialog box opens. The **Profiles** theme is a polygon theme containing the defined profiles and the information about each of the profiles.



### 7.2.1 Recommended procedure

#### Define the profiles

- Specify the bandwidth and the overlap. If a negative overlap is specified the profiles are made with space in-between.
- Click on the **N-S profiles** or the **E-W profiles** check box to generate the profiles automatically. Click off the checkboxes to delete the profiles.
- Press the **Manual** button to define the profile manually, double-click to end a profile.
- Press the **Snap to Borehole** button to define the profile manually while snapping to the boreholes.
- Use the **Offset** tools to offset the current profile. The current profile is the selected profile.

#### Select and edit the profiles

- Select the profiles by either of two ways:
  - Select the profile by ID using the **Select by ID**
  - Select the profile by using the **Select by clicking** tool and click within a profile.
- When a profile is selected the boreholes within the profile are also selected and the projection lines to the profile line are drawn. When the user selects a profile the profile will be moved to the bottom of the attribute table so that the profile will appear on top of the other profiles.
- Use the **Change ID** button to change the id of a selected profile.
- Use the **Remove Profile** button to delete a selected profile, in the **Model-area** view.
- Use the **Select All** button to select all the profiles. Using the **Remove Profile** option while all the profiles are selected removes all the profiles.
- Use the **View Current profile** to view the vertical view for the current profile.
- Use the **View All Profiles** to view the vertical views for all the profiles.



### 7.2.2 Content of the profile definition dialog box

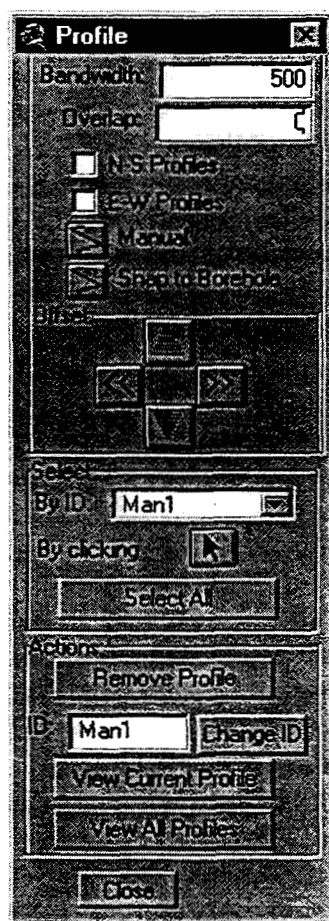


Figure 11 Profile Definition Dialog Box.

**Bandwidth:** Sets the bandwidth for the profiles.

**Overlap:** Sets the overlap for the profiles. If the overlap is negative there will be space between the profiles.

**N-S profiles:** Clicked on, n-s profiles covering all boreholes are generated according to the specifications in the **Bandwidth** and the **Overlap** box. Clicked off again all n-s profiles are deleted.

**E-W profiles:** Clicked on, e-w profiles covering all of the boreholes are generated according to the specifications in the **Bandwidth** and the **Overlap** box. Clicked off again all e-ws profiles will be deleted.

**Manual:** Select this tool for defining a profile. Click on the screen to start a profile, double click to end.



**Snap to Borehole:** Define profile by clicking while snapping to the nearest borehole.

**Offset profile:** offset the current profile, using the overlap defined in the **Overlap** box.

**Select Profile:** When the profiles are defined, the user can select a profile in two ways;

- 1) Press the tool-button (**Select by clicking**) and then click inside a profile.
- 2) Select by ID.

When the user selects a profile, all the wells inside the profile are also selected, and the projection lines between the wells and the profile line are drawn.

**Select All:** Select all the profiles.

**Remove:** This will delete the current profile. If all the profiles are selected they are all removed.

**Change ID:** Enables the user to change the id for the selected profile. This could be used to add a special comment to the profile. Every time a new profile is created it will, by default, get an ID:

- 1) For **Manual** or **Snap to Borehole** generated profiles, Man + a number.
- 2) For automatically generated N-S profiles, NS + a number.
- 3) For automatically generated E-W profiles, EW + a number.

Typing a new ID in the id-text box and pressing the button changes the ID. The GeoEditor will check if the new ID already exists, if so the user will not be able to change the ID.

**View current profile:** Show the vertical view for the current profile. When viewing the profiles the user is only able to view and not define the geology.

**View all profiles:** Show the vertical views for all the profiles.

### 7.3 Interpret Geology

When the user has defined the profiles, see Section 7.2, and has defined any TEM or layer data to be included in the vertical profiles, see Chapter 12, the user is ready to start the geological interpretation.

Generating a vertical view for each of the defined profiles does the geological interpretation. The vertical views are either generated by using the **View all profiles** option in the **Profile** dialog box or using

275



the **Digitise Layers** options in the **develop Geological Model** dialog box.

When the GeoEditor has generated the vertical views, they will appear on the lower 2/3 of the screen. The upper 1/3 of the screen will contain the horizontal view, which will be zoomed in to the currently selected profile. The view will also contain the **Digitise layers** dialog box, which is the main dialog box when interpreting the geology, see Figure 12.

### 7.3.1 Recommended procedure

#### Define layers

- First define the layers. The Define layers dialog box is accessible through the Define Layers dialog box, see Section 7.4.

#### Digitise layers

- Select the profile to work with by selecting a profile by ID or by clicking.
- Select the layer to digitise from the **Select layer** list in the **Digitise layers** dialog box.
- Use the **Edit tools** to digitise the layer, see Section 7.5.
- Select a new layer from the **Select layer** list in the **Digitise layers** dialog box.
- Use the **Remove profile** button to remove profiles not yielding any information for the present interpretation. See Section 7.5.
- Use the **Add Borehole** to add a new borehole. See Section 10.2.
- Use the **Mark area for close-up view** to view boreholes drawn on top of each other.
- Use the **Schematic view** tool to see the horizontal position and ID for the boreholes..
- Use the **Add layer or Tem data to view** option in the **right mouse popup menu** , to add grid data or TEM data to the view. See Chapter 12.
- Use the **Close** button to return to the horizontal view.

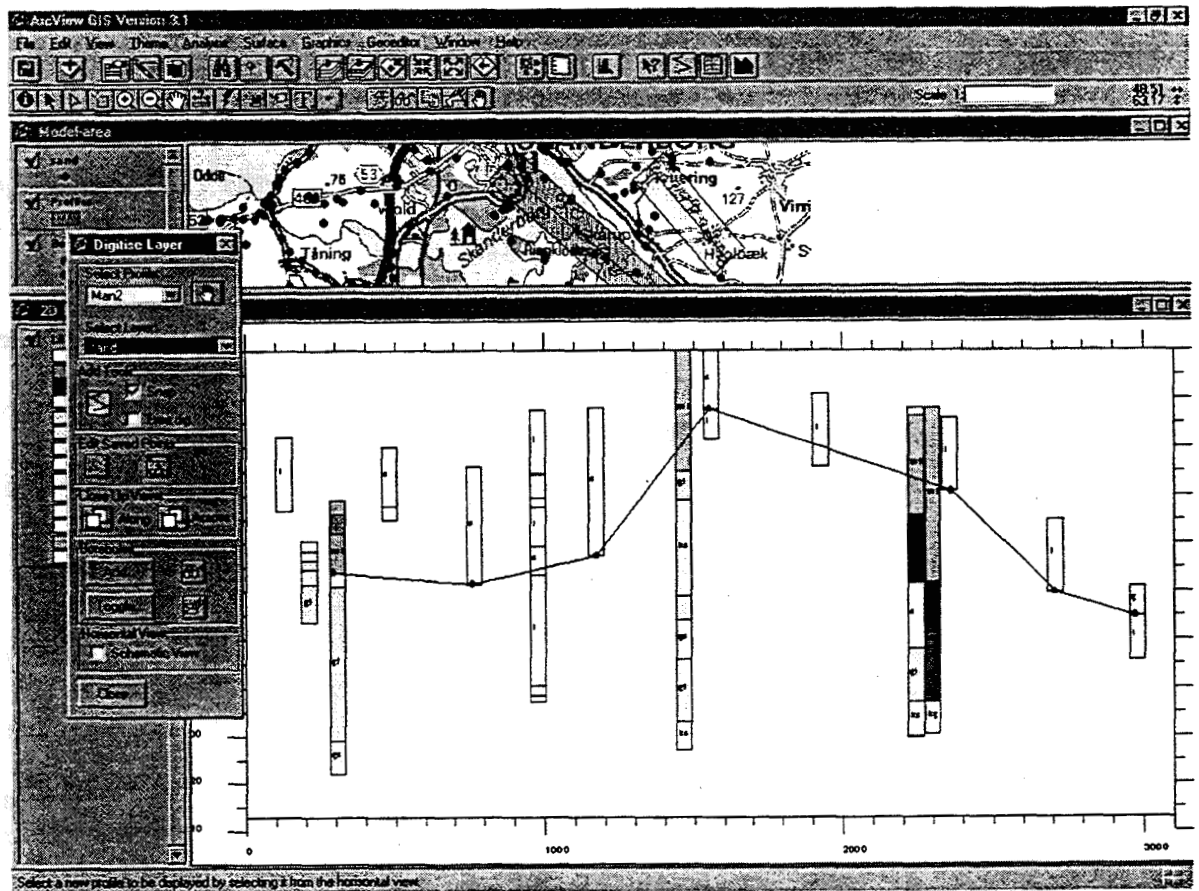


Figure 12 The View when Interpreting the Geology.

## 7.4 Define Layers

The layers used in the GeoEditor are defined as point themes in the horizontal view, as the layers are defined by discrete points. When finished with the geological interpretation the discrete points for each layer can be interpolated to a spatial map, for validation or export.

The layers are defined in the **Define layers** dialog box, which are accessible from the **Develop Geological Model** menu under **Define Layers**. The user can define a layer from both the vertical and the horizontal view. By defining the layers means that for all layers the lower border is defined.

When defining a layer a point theme, having the same name as the layer, is generated and shown in the horizontal view. When the user then defines discrete values for that layer the points will be saved to the point theme and shown on the horizontal view. This will enable the user to see the horizontal location of the discrete values when they are digitised in the vertical view.





#### 7.4.1 Recommended procedure

- Type a layer name in the **Layer name** box.
- Select a colour for the layer in the **Choose layer colour** list.
- Press the **Define layer** button, and the layer will be added to the list below, and a point theme with the layer name, will be added to the horizontal view.
- Toggle the layer ordering by using the **Up** or **Down** tool.
- To load a layer from another GeoEditor project, use the **Load layer** button.
- To remove a layer from the list, select the layer and press the **Remove layer** button.
- To remove all the defined layers, press the **Clear all** button.

#### 7.4.2 Content of the define layers dialog box

**Layer name:** Type the name of the layer. The user should avoid to use 'special' characters, e.g.: **\_**, **%**, **#**, in the layer name as it can give problems when converting the layer theme to an interpolated surface.

**Lens:** If the **Lens** option is clicked on. There will be generated two themes; an upper and a lower theme. When working with lenses it is necessary to define both the upper and the lower border. For all other layers only the lower border is defined.

**Choose a layer colour:** Choose the colour of the layer. This colour will follow the theme when it is presented. The colour could be changed manually by editing the legend for the associated point theme.

**Define layer:** When the user presses this button the layer is added to the list below, and a point theme, having the same name as the layer, is defined. The layer name will also be added to the **Select layer** list in the **Digitise layer** dialog box.

**Up and Down tool:** The up and down tool toggles the layer order. The layer order is used whenever the layers are presented so that the top layer is always shown at the top. The layer order is also adopted when validating the output.

**Load layer:** This enables the user to load an already defined layer from an external point theme. Although it is required that the point theme is generated by the GeoEditor.

**Remove layer:** To remove a layer from the list, select the layer and press the **remove layer** button. The point theme in the horizontal view is also deleted at the same time. The user is asked if the entire defined



layer borders, in all the profiles, belonging to the removed layer should be removed.

**Clear all:** This clears all the defined layers.

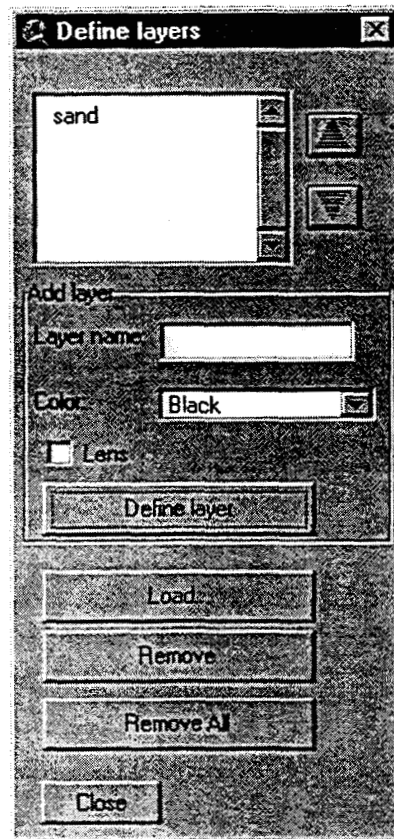


Figure 13 Define Layers Dialog

## 7.5 Digitise Layers

Using the Digitise layers dialog box, which is opened automatically when generating a vertical view, does the definition of the layer borders.

Before digitising the layer borders begins, the user must have generated some vertical views, see Section 7.2 , and defined one or more layers, see Section 7.4.

### Recommended Procedure

- Define the layers. See Section 7.4.
- Select the profile to work with.
- Select the layer to digitise from the **Select layer** list.
- Use the **Snap** option to snap to nearest border and the **Free dig** to define points outside the boreholes.
- Add new points using the **Add new point** tool.



- Use the **Change position** or **Delete point** tool to edit the points.
- Use the **Mark area for close-up view** button to view any overlapping boreholes, along or across profile.
- Use the **Remove Boreholes** to remove boreholes from the view.
- Use the **Add Borehole** to add a 'ghost' borehole to the view.
- Select a new profile by using the **Select new profile** tools.

### 7.5.1 Content of the digitise layers dialog box

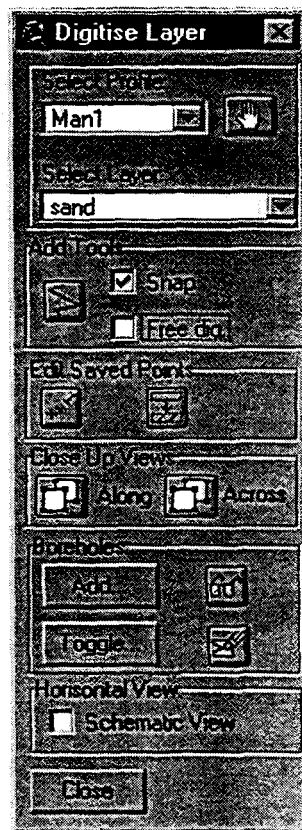


Figure 14 Digitise Layers Dialog.

**Select new profile:** If several vertical views are created there are two different ways to toggle between the views:

- 1) Use the **Select profile** option and select from the list the vertical view to be viewed.
- 2) Use the **Select on view** option, and select from the horizontal view profile to be viewed.

**Choose layer:** All the defined layers will be listed here. The user should select the one layer to work with. When a layer is selected the edit tools will be enabled. To select a layer the desired layer in the listbox is clicked on. The GeoEditor will check if the layer contains



some discrete points defined for boreholes included in this profile. In case the digitised values will appear on the screen with a small green line indicating already defined values.

Also the selected layer will, by default, be the only visible layer theme in the horizontal view.

When the user selects a layer the box will be enabled.

**Snap:** This will snap the point to the nearest border, when clicking within borehole. When clicking outside a borehole the snap function will not be used.

**Free dig.:** This will allow the user to define points outside the boreholes. If this is clicked off the user is not allowed to define a point outside a borehole. When clicking outside a borehole the points are located at:

- a) If there are two neighbouring boreholes, at the connection line between them.
- b) If there are only one neighbouring borehole the new point will be located at the profile line.

**Add new point:** This tool is enabled when a layer is selected. Click the tool and click on the screen to define the layer points, only one click per point. When clicking on the screen lines will connect the points. Newly defined points are marked with a red mark, when the point is saved it will be marked as a black point. The points are saved automatically by the system.

**Change Position:** Changes the position of a defined point. This option only works for points defined within a borehole, when clicking at a new location within the borehole, the point connected to this borehole is moved to the new location.

**Delete Point:** Deletes a defined point. Click on the point to delete it. This option works for all points.

**Borehole information tool:** When clicking on a borehole in the vertical the borehole information is shown, see Chapter 11 for further description.

**Add Borehole:** This will add a new borehole to the vertical view. See Section 10.2.

**Remove well:** The remove borehole option allows the user to remove a borehole from the vertical view that doesn't contain any relevant information for the present interpretation. Removing a borehole only excludes it from the currently profile, and not from the database or from any other profiles it may be located in. When the user clicks within a borehole profile the user is prompted if the selected borehole



should be deleted. If the user answers yes the borehole profile is removed, and the tool remains active. If the user answers no the borehole profile is not removed and the tool remains active. If the user answers cancel the tool is made inactive.

**Mark area for close-up view:** When the **Show all with true position** option is used in the **View settings** dialog box, see Chapter 12. The boreholes are showed at their true position, this means that several boreholes could be placed on top of each other, as they are projected on to the profile line. The tool allows the user to see the boreholes either **along profile** or **across profile**. Select the tool and draw a rectangle on the view, to indicate which boreholes to include. If the **along profile** option is selected the boreholes will be displayed beside each other in a small view. If the **across profile** option is selected the boreholes will be shown with their position in relation to the profile line. When the small view is closed it will be deleted automatically. The user is able to delete the boreholes and to get information using the appropriate tools in the close-up view.

**Schematic view:** When this is clicked on a view containing a graphic showing the profile line and the position of the boreholes, with their ID, will be shown in the upper 1/3 of the screen. Clicking the check box of deletes the view.

**Toggle:** Toggle the borehole visibility on and off.

**Close:** Close the vertical view and return to the horizontal view.



## 8 DEPTH INTERVAL

With this utility it is possible to develop surface maps representing zonations of characteristic aquifer properties.

The Depth Interval mode is entered from the **Model approach dialog box**, see Chapter 6, or via the Settings menu during a **Vertical Profiles** session, see Section 12.3.

The purpose of the Depth Interval approach is to define zonations of characteristic aquifer properties for a specified depth interval. In contrast to the other geological approach presented in this manual, see Chapter 7, the geology is classified after the dominant lithology in a specified horizontal layer.

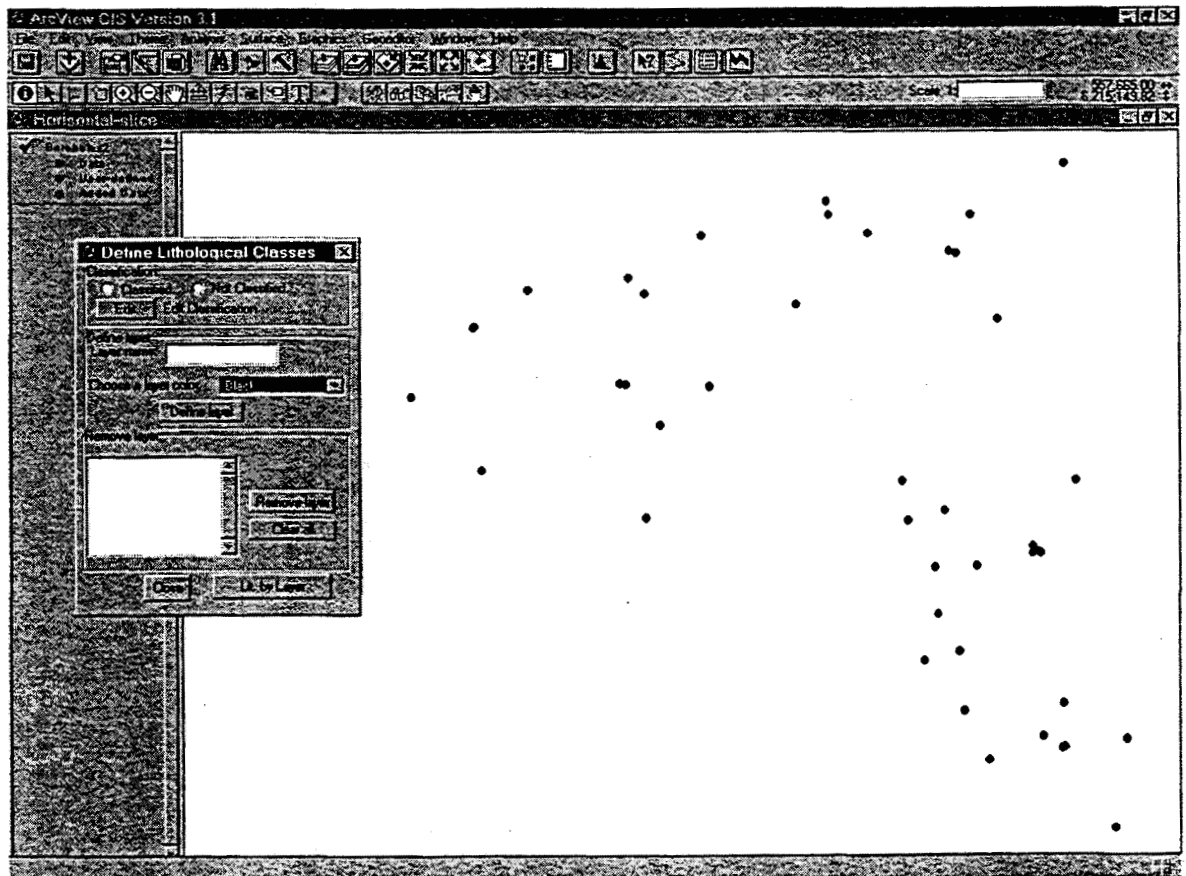


Figure 15 The Horizontal-slice View.

When entering the Depth Interval mode a new window, the **Horizontal-slice** view, will be created including a theme, **Boreholes2**, containing the selected boreholes, see Figure 15.



The procedure for this approach is:

- Specify if the lithology should be classified or not.
- If classified is selected: Classify the lithology in groups. These groups will act as layers representing the different zonations.
- If the lithology is selected not to be classified: Define the layers representing the different zonations.
- Specify the depth intervals from which the dominant lithology should be determined.
- Define the zonations by automatic fill (only if the lithology was classified) or manually.
- Edit the defined zonations.
- Convert the maps of the zonations to grids.
- Edit and validate the grids.
- Export the grids.

## 8.1 Define Lithological Classes

When selecting the "Depth Interval" option, a new view "Horizontal-slice" is created, and the "Define Lithological Classes" dialog box is opened, see Figure 16. Here the user can choose to classify the lithology in up to five groups or define layers. If the user classifies the lithology in groups these groups will act as layers, otherwise a number of layers has to be defined. Each of the defined layers will represent one zonation.

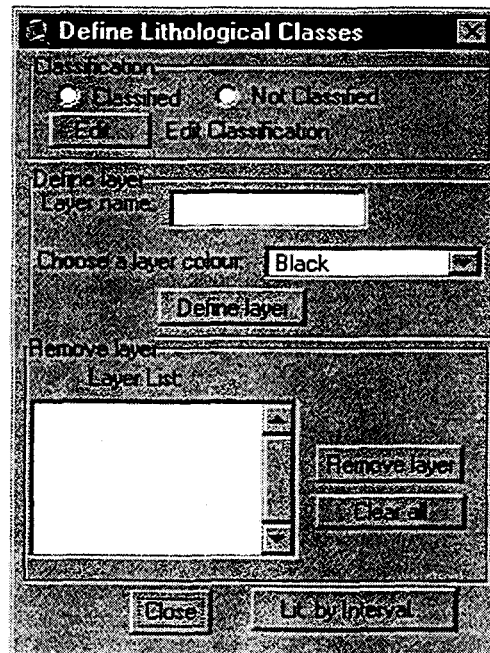


Figure 16 Define Lithological Classes Dialog.



### 8.1.1 Recommended procedure

- Select whether the lithology should be classified or not.

#### If classified is selected.

- Select the **Edit** button to open the **Classification of Lithology** dialog box, see Section 8.1.4 for a more detailed description of this dialog box.
- Select the number of classes to include, by toggling the **visibility checkbox** on and off. By default all five classes will be selected initially.
- Change the name of the classes by typing a new name in the **name box**.
- Remove a lithology by selecting it from the list and pressing the **Remove** button.
- Add a lithology to a group by;
  - a) Selecting the new lithology from the **Add list**.
  - b) Activate the group by clicking in the **lithology list**.
  - c) Press the **Add** button.
- Press the **Close button** to close this dialog box and return to the **Define Lithology Classes** dialog box, see Figure 16.
- Press the **Lit. by Interval** button to close the **Define Lithology Classes** dialog box and open the **Lithology by Interval** dialog box, see Figure 18.

#### If not-classified is selected.

- Define a new layer by typing a name in the **Layer name box**.
- Choose a layer colour representing this layer.
- Press the **Define layer** button, and the layer will be added to the **Layer list**.
- To remove a layer from the Layer list, select the layer and press the **Remove layer** button.
- To remove all the layers from the Layer list, press the **Clear all** button.
- Press the **Lit. by Interval** to close the **Define Lithology Classes** dialog box, and open the **Lithology by Interval** dialog box, see Figure 18.

### 8.1.2 Content of the define lithological classes dialog box

**Classified:** Select this option if groups should classify the lithology. Then the controls in the lower part of the dialog box specifying the layers will be disabled. This option typically will be used if the geology in the area is very inhomogeneous and contains many different formations.

285





**Not Classified:** Select this option if the lithology should not be classified. This option is useful if the geology in the area is rather homogeneous and only contains a limited number of formations.

**Edit:** This will open the **Classification of Lithology** dialog box, where the user should specify how the lithology should be grouped.

The rest of the controls will only be enabled if the user selects the **Not Classified** option.

**Layer Name:** Type the layer name. Each layer will later on represent a specific zonation.

**Choose a layer colour:** Choose a layer colour for the layer.

**Define Layer:** Press to generate the layer. Then the layer will be added to the list at the bottom of the dialog box.

**Remove layer:** Select a layer from the list. Press the **Remove** button to remove the layer from the list.

**Clear all:** Clears all the layers from the list.

**Lit. by Interval:** Opens the lithology by Interval dialog box

### 8.1.3 Not classified

If the **Not classified** option is selected the user should define layers representing each zonation of a specific geological layer. When the dominant lithology in the user specified layers is calculated, see Section 8.2, the lithology will not be classified. If the geology in the model area is very inhomogeneous and contains many different geological formations, it could be difficult to get an easy overview of the data. Thus it would be preferable to classify the lithology. If the geology in the model area on the other hand is rather homogenous and only contains a limited number of geological formations, the **Not classified** option would be preferable.

### 8.1.4 Classified

If the **classified** option is chosen the **Edit** button becomes enabled. Pressing the **Edit** button opens up the **Classification of Lithology** dialog box, see Figure 17.

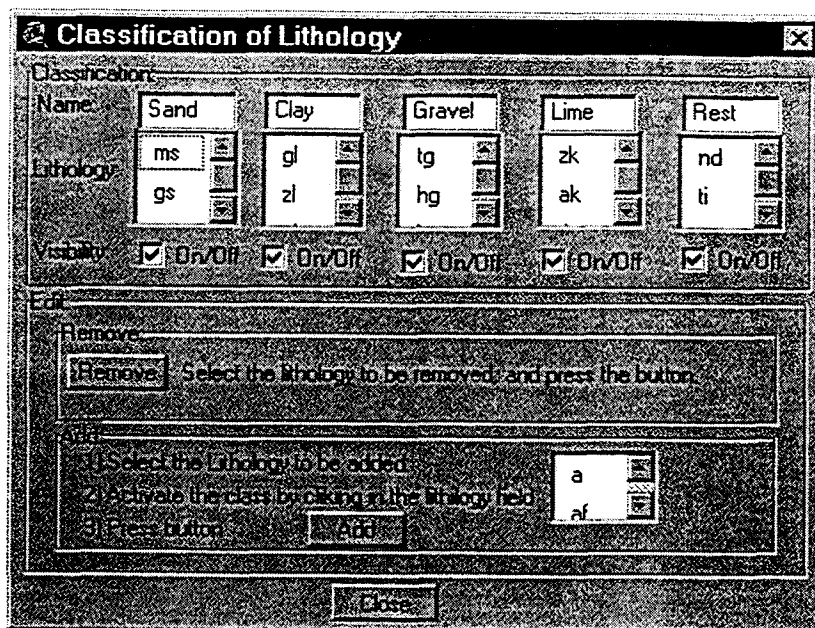


Figure 17 Classification of Lithology Dialog.

The classification of lithology in groups is preferable when there are many geological formations in the area. By classifying the lithology in groups the geology will be simplified and it will be easier to get an overview of the geology in the area.

### 8.1.5 Content of the classification of lithology dialog box

**Name:** Type the name of each group.

**Lithology:** Contains the lithology symbols for each group.

**Visibility:** Checked on if the group is used.

**Remove:** Remove a lithology from a group by selecting the lithology in the list, and pressing the **Remove** button.

**Add:** To add a new lithology to a group: select the symbol from the list at the bottom of the dialog box. Activate the group by clicking on the list, then press the **Add** button.

**Close:** Return to the **Define Lithological Classes** dialog box, see Figure 16.



## 8.2 Lithology by Interval

When the user has defined the classification type, whether it is classified or not, and the layers, the next step is to define the intervals from which the dominant lithology should be calculated.

The intervals are defined in the **Lithology by Interval** dialog box, see Figure 18. This dialog box could be reached by pressing the **Lit. by Interval** button in the **Define Lithological Classes** dialog box.

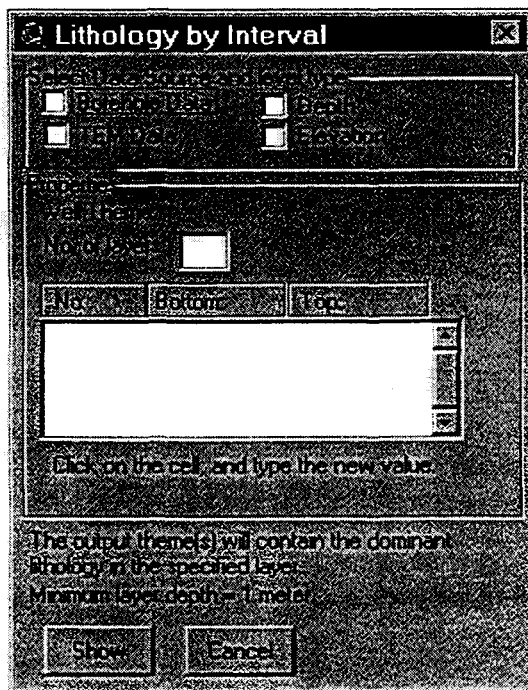


Figure 18 Lithology by Interval Dialog.

### 8.2.1 Recommended procedure

- Select whether the source theme is a borehole theme or a TEM theme.
- Select if the interval should be specified as metre below surface (depth), or as metre above sea level (elevation).
- Define the number of intervals. For each interval the dominant lithology for each borehole will be calculated and displayed in a theme. The output will be a polygon theme for each interval.
- Specify the bottom and top elevation for each of the intervals.
- Press the **Show** button to calculate the dominant lithology. This will close the **Lithology by Interval** dialog box and open the **Define Zonations** dialog box.



### 8.2.2 Content of the lithology by interval dialog box

**Select Data Source and level type:** Select whether the data theme is a borehole theme or a TEM theme. Then specify whether the elevation should be specified as depth, metre below surface, or as level, metre above sea level. Using the level type the user can specify whether the interval should follow the surface or it should follow some specified elevations.

**Properties:** Define number of depth intervals, for each depth interval a theme containing the dominant lithology is generated. For each interval the bottom and the top level should be specified. A dominant lithology is only included if the depth of the layer is more than one metre.

**Show:** Extracts the dominant lithology for each defined interval, and opens the **Define Zonations** dialog box. If the classified option was used the theme containing the dominant lithology will be classified after the defined groups. There will be two options for defining the horizontal extension for each group, a) manually definition or b) automatically definition. If the **Not Classified** option is used only the manually definition is available.

## 8.3 Specifying the Zonations

The **Define Zonations** dialog box is used to define the zonation of the geology in the model area. This dialog box is opened automatically when the user presses the **Show** button in the **Lithology by Interval** dialog box, or it could be accessed from the main menu by selecting **2.3b Define Zonations**.

For each defined interval where the dominant lithology is calculated for each borehole, the output will be two themes;

- 1) A point theme, Lit.: top – bottom m.b.s or m.a.s.l. (metre below surface or metre above sea level) showing the location of the boreholes and the dominant lithology for this borehole in the specified depth interval.
- 2) A polygon theme, Layer top – bottom, use to define the zonations for this interval.

The purpose of this dialog box is to convert the calculated point theme, showing the dominant lithology for each borehole in the specific depth interval into zonations.

This could be done automatically, if the lithology is classified, or manually.



### 8.3.1 Recommended procedure

#### For automatic zonation

- First define the grid, by pressing the **Grid Set-up** button. This will open the **Grid Set-up** dialog box. See Section 9.2.
- Select whether the zonations should be defined by **Theissen** polygons or by **Neighbourhood**.
- Press the **Fill** button to activate the automatic zonation.
- The output will be a point theme showing the codes for each of the boreholes and a grid theme containing the calculated zonations.
- When using the automatic zonation the output will be a grid. Press the **Convert to Grid** button to open the **Convert to Surface** dialog box, see Section 9.3. This will enable the user to validate the surface, before exporting.

#### For manual zonation

- Select the interval to edit, by selecting a theme from the **Select theme** list.
- To define a zone;
  - a) Select a layer from the list (cursor in cross mode)
  - b) Define the zones by clicking on the screen, double click to end.
- Delete a zone by pressing the **Delete** button and clicking inside the polygon.
- Edit a zone by pressing the **Edit Shapes** button. The polygons are editable by selecting the **Edit Shapes** button and then clicking inside the polygon to edit. When selecting a polygon handles will appear in all the corners. New handles will appear if clicking at a boundary. The handles are moveable in order to change the shape of the polygon.
- Press the **Save** button to save changes.
- Press the **Edit Zonation** button to open the **Edit Selected Zonation** dialog box, see Section 8.3.5. This requires a predefined grid. This enables the user to specify a specific layer on a cell to cell basis.
- Press the **Convert to Grid** button to open the **Convert to Grid** dialog box, see Section 9.3. This will enable the user to convert the defined zonation to grid, and to validate the grid.

### 8.3.2 Content of the define zonations dialog box

**Grid set-up:** Opens the **Grid Set-up** dialog box see Section 9.3. Prior to using the automatic option it is necessary to define the model grid, as the procedures only works on grids. If the user wants to use the manual option it is NOT necessary to predefine a grid first.

**Fill by Theissen Polygons:** This option is enabled if the user has classified the lithology, and if there exists a predefined grid. The



interval theme to be converted by Theissen polygons has to be the only active theme. The output will be a point theme containing the locations and the codes for each of the groups and a grid theme containing the Theissen polygons. When converting by Theissen polygons all groups are assigned a number.

**Fill by Neighbourhood:** This option is enabled if the user has classified the lithology and if there exists a predefined grid. The interval theme to be converted by Neighbourhood has to be the only active theme. The output will be a point theme containing the locations and the codes for each of the groups and a grid theme containing the zonations. When converting to zonations all groups are assigned a number.

**Fill:** Press this button to activate the automatic fill.

**Select Theme:** This list will contain the themes for each defined interval. When selecting a theme in the list the associated polygon theme, used in the manual zonation, is activated. The working interval should be selected from this **Select Theme** list when using the manual zonation.

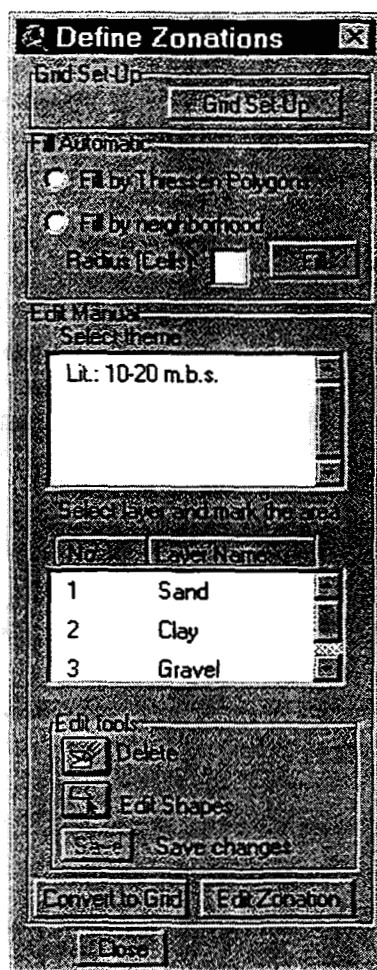


Figure 19 Define Zonations Dialog.

**Select layer and mark the area:** This list will contain the layers/zones to edit/generate. If the lithology is classified the defined groups will appear here. If the lithology is not classified the defined layers will appear here. When using the manual zonation, the cursor will change to cross mode when selecting a layer. Now the user can define a polygon by clicking on the screen, double-click to end.

**Delete:** Remove a polygon by clicking the **Delete** button, and clicking inside the polygon to be deleted.

**Edit Shapes:** This tool only works if the zonations are defined manually. The polygons are editable by selecting the **Edit Shapes** button, and then clicking inside the polygon to edit. When selecting a polygon handles will appear in all corners. New handles will appear if clicking at a boundary. The handles are moveable in order to change the shape of the polygon.

**Save:** Press the **Save** button to save the changes made by the **Edit Shapes**.



**Edit Zonation:** Opens the **Edit Selected Zonation** dialog box, see Section 8.3.5. This requires a predefined grid. The ability enables the user to specify a specific layer on a cell to cell basis.

**Convert to Grid:** Opens the **Convert to Grid** dialog box see Section 9.3. If the automatic generation was used, a grid has already been created. Hence there will be no need of **Convert to Grid**.

### **8.3.3 Automatic generation of layer extension**

If the classified option was used, the theme containing the dominant lithology will be classified after the defined groups with a maximum of five groups. The extension of each group within the depth interval could be calculated using the **Theissen Polygon** option or the **Neighbourhood** option as the output from these interpretations would be a surface map. The model grid should be defined in advance. Press the **Grid set-up** button to open the **Grid Set-up** dialog box, see Section 9.2.

### **8.3.4 Manually generation of layer extension**

BY the manual edition of the layer boundaries the user defines the boundaries as polygons, where each of the layers are assigned a number.

### **8.3.5 Edit zonation**

Pressing the **Edit Zonations** button in the **Define Zonations** dialog box, opens the **Edit Selected Zonation** dialog box, enabling the user to specify a number on a per cell basis. Prior to using this option the user will have to define a grid as the grid set-up is used to define the cells.

The user assigns a number to a cell by selecting a layer from the list, pressing the tool, and clicking on a cell within the grid. Thus the cell will be assigned the number corresponding to the selected layer.

293







## 9 VALIDATION AND EXPORT

The validation and export is accessible through the **Validation and Export** option under the main menu. The validation and Export dialog box enables the user to select different validation options.

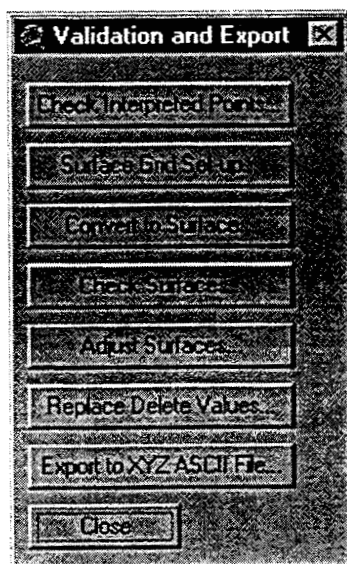


Figure 20 Validation and Export Dialog Box.

### 9.1 Check Interpreted Points

This option validates the interpreted points, meaning that it checks for overlapping layers using the user specified layer order.

When selection the **Check Interpreted Points** option the **Check Interpreted Points** dialog box appears. The dialog box displays the defined layers in the same ordering as they are defined in. The layers are checked for overlapping layers using the specified layer order, the output from the check is a point theme for each layer check showing the location of any errors. In case of errors the user should open the vertical view and move the digitised point, so that overlapping layers are avoided.

#### 9.1.1 Recommended procedure

- Use the **up** or **down** button to change the order of the layers.
- Use the **Add** button to add a new layer to the list.
- Select a layer and press the **remove** button to remove a layer.
- Press the **Check** button to execute the check.



### 9.1.2 Content of the check interpreted points dialog box

**Up and down button:** Moves the selected layer up or down in the ordering. When the dialog box opens the layers are ordered according to the order defined when defining the layers.

**Remove:** Removes the selected layer from the list.

**Add:** Adds a new point theme to the list. When checking the layers the point theme should contain a z-field.

**Check:** Executes the check. The layers are checked two and two. If a layer is ok, the message box will display "layer1-layer2 OK". If any overlapping points were found the message box will display "layer1 - layer2 crossing in XX boreholes", and a point theme showing the location of the overlapping points is added to the view. To edit the overlapping points the user has to open the vertical views and move the points.

**Close:** Closes the dialog box.

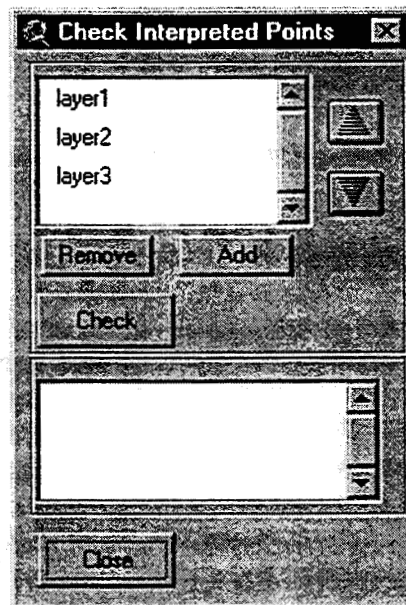


Figure 21 Check Interpreted Points Dialog.

## 9.2 Surface Grid Set Up

This menu allows definition of the model area and the model grid, see Figure 21. The model grid is used when converting themes to surface grids.



### 9.2.1 Recommended procedure

- Select the **Grid on/off** to display the grid.
- To change the grid geometry type new values in the fields of the **Set Up**. Select the **Grid on/off** to display the revised grid.
- Click the **Load Grid** to import the grid set-up from an external file. The supported files are T2 file and DHI grid set-up ASCII file, the formats are described in Appendix 2
- To save a grid set-up to an ASCII file, select the **Save grid**. The format of the grid definition ASCII file is described in Appendix 2.
- Press the **Convert to Surface** button to open the **Convert to Surface** dialog box, see Figure 22.

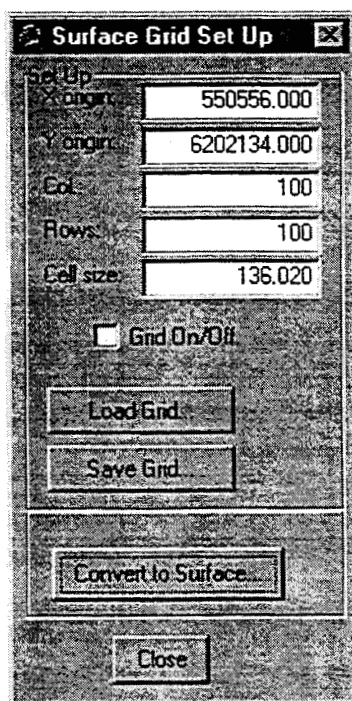


Figure 22 Surface Grid Set Up Dialog.

### 9.2.2 Content of the grid set up dialog box

**Set Up:** Define the geometrical parameters of the grid by typing the values. When the dialog box opens the default origin will be taken from the active theme. The number of columns and rows is always 100 per default. The cell size will be calculated accordingly. Use the **Grid on/off** to show the grid.

**Load Grid:** Opens a browse dialog box for selecting a grid file. The supported files are T2 file and DHI grid set-up ASCII file, the formats are described in Appendix 2

296



**Save Grid:** Save the set-up values to an ASCII file. The format of the grid definition ASCII file is described in Appendix 2.

**Grid on/off:** Turns the grid on and off.

**Model area on/off:** Turns the model area on and off.

**Convert to Surface:** Opens the convert to surface dialog box, see Section 9.3.

## **9.3 Convert to Surface**

This menu allows the user to define the themes that should be converted to surfaces using the defined surface grid set-up, see Figure 22.

### **9.3.1 Recommended procedure**

- Use the **Add** button to add themes to the list.
- Use the **Remove** button to remove the themes that should not be converted to grid.
- Press the **Convert to Surface** button to convert the themes in the list to grid themes.

### **9.3.2 Content of the convert to surface dialog box**

**Add:** Add themes to the list

**Remove:** select a theme in the list and press the remove button to remove the theme from the list.

**Convert to surface:** Converts all the themes in the list to surfaces (ArcView grids) using the model grid definition from the surface grid set-up, see Figure 22. Prior to using this option the surface grid set-up must be done. Uses a default interpolator which is Inverse Distance Weighted (IDW) with a power of 2, no barriers, and a variable Radius of 12 points with no maximum distance.

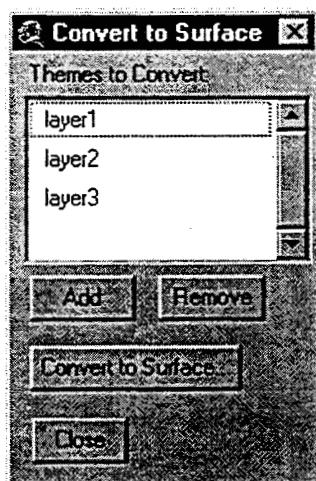


Figure 23 Convert to Surface Dialog.

## 9.4 Check Surfaces

This dialog box enables the user to check surfaces against each other for overlapping layers.

When checking for overlapping layers the surfaces have to be arranged in a descending order with the grids representing the upper layers above the grids representing the lower layers.

When pressing the **Check** button the layers, represented as surfaces, will be checked for overlapping values. Errors will be displayed in an **Error message box**.

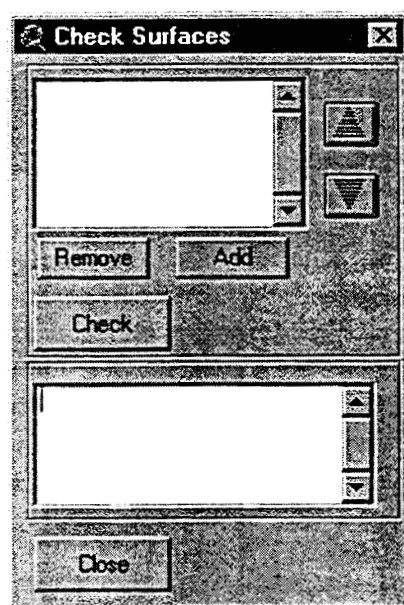


Figure 24 Check Surfaces Dialog Box.



#### 9.4.1 Recommended procedure

- Use the **up** or **down** button to change the order of the selected surface.
- Use the **Add** button to add a surface to the list.
- Select a surface from the list and press the **Remove** button to remove it from the list..
- Press the **Check** button to execute the check.

#### 9.4.2 Content of the check surfaces dialog box

**Up and down button:** Moves the selected surface up or down in the ordering. When the dialog box opens the surfaces are ordered according to the order defined when defining the layers.

**Remove:** Removes the selected surface from the list.

**Add:** Adds a new surface to the list.

**Check:** Executes the check. The surfaces are checked two and two. If a surface is ok, the message box will display "surface1-surface2 OK". If any overlapping points where found the message box will display "surface1 - surface2 crossing", and a surface showing the thickness of the layer is added to the view.

### 9.5 Adjust Surfaces

This menu allows the user to adjust a surface according to a reference surface, e.g. the surface should be X meters below the reference surface in a specified area, or all overlapping values should be X meters below the reference surface.

#### 9.5.1 Recommended procedure

- Specify the **Upper** and the **Lower** surface.
- Specify the reference surface. The reference surface is NOT adjusted.
- Specify the minimum layer thickness.
- Specify the areal extent. If the areal extent is not specified the entire area is adjusted.
- Specify if the whole area should be adjusted or only areas with overlapping values.
- Press the **Adjust** button to execute, the output is a surface containing the adjusted values.



### 9.5.2 Content of the adjust surface dialog box

**Upper and Lower Surface:** Specify the upper and lower surface, this information is used when adjusting the surface.

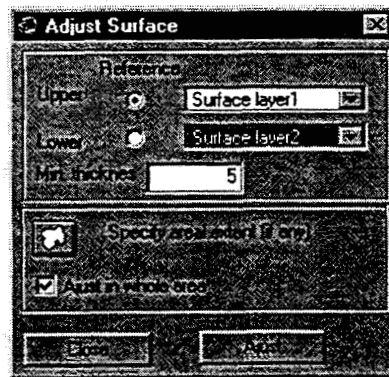


Figure 25 Adjust Surface Dialog Box.

**Reference Surface:** Specify the reference surface. The reference surface is not adjusted, but is used as reference for the other surface.

**Minimum thickness:** Specify the minimum layer thickness, when adjusting the surfaces the minimum thickness is used to define the maximum adjustment.

**Areal Extent:** By default the adjustment is done for the entire surface area, but the user has the option of specifying an areal extent for the adjustment.

**Adjust in whole area:** By default the adjustment is only done for areas with overlapping values (negative layer thickness). If the adjustment should be done in the entire area the **Adjust in whole area** should be checked on.

## 9.6 Replace delete Values

Enables the user to replace delete values in a surface grid with a specified value.



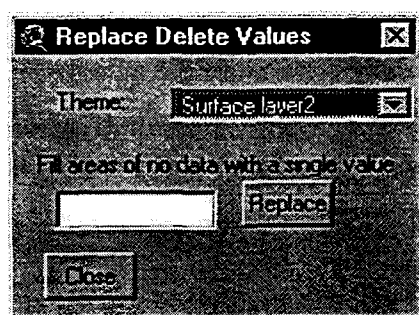


Figure 26 Replace Delete Values Dialog Box.

### 9.6.1 Recommended procedure

- Select the surface theme from the list.
- Specify the single value to replace the delete values.
- Press the **Replace** button to execute the **Replace Delete Values**.

## 9.7 Export to XYZ File

This feature allows the user to export a point theme or a surface theme to a XYZ ASCII file. The feature could be useful if the user wants to utilise information of the GeoEditor in other applications.

### 9.7.1 Recommended procedure

- Select export of point theme or export of grid theme.
- Select the theme from the list. In case of export of a point theme do select the field containing the z-values.
- Specify the output file by typing the path or by using the browse button.
- Press the **Export** button to start the export.

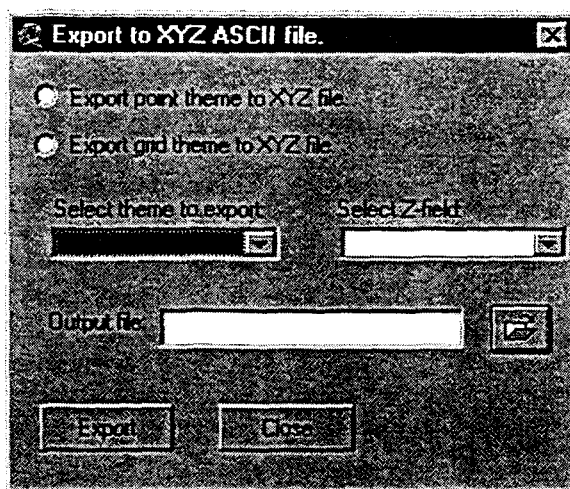


Figure 27 Export to XYZ ASCII File.

### 9.7.2 Content of the export to XYZ file dialog box

**Export point theme to XYZ theme:** Select this option to export a point theme to XYZ file.

**Export grid theme to XYZ theme:** Select this option to export a grid theme to XYZ file. The XYZ file will contain the X- and Y-coordinates of the centre of each cell.

**Select theme to export:** Selects the theme to be exported. The content of the list will reflect the specified export type. If e.g. a point theme is selected only the point themes of the active view will be displayed in the list.

**Select Z-field:** If the point theme option is selected the Z-value should be chosen from the list. Only the numerical fields of the point theme will be listed.

**Output file:** Specify the output ASCII file. Type the filename or use the browse button.

**Export:** Starts the export.

**Close:** Closes the dialog box.





## 10 ADD DATA TO THE PROJECT

In the process of doing a geological interpretation it can be useful/necessary to validate the information from the boreholes with other data. In the GeoEditor the user has the possibility to geophysical data as TEM (Transient Electro-Magnetic data) or geoelectrical data. The GeoEditor supports adding a single boreholes or a number of boreholes from a database as well.

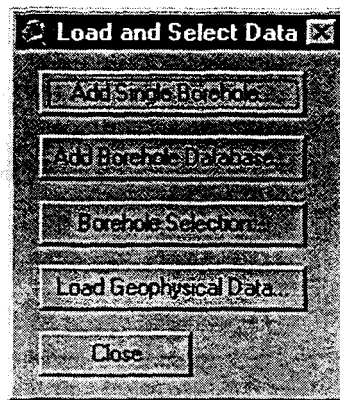


Figure 28 Load and Select Data Dialog Box.

When the user selects **Load and Select Data** from the **GeoEditor** menu the following dialog box appears.

### 10.1 Add GIS Data

Spatial data in GIS format is added using standard ArcView® procedures, see the ESRI® manuals for a further description.

### 10.2 Add Single Borehole

The option for adding a new borehole enables the user to add a single borehole if e.g. external data indicate a certain geological layer in that area or e.g. to include a newly constructed borehole to the project. The add new borehole is assessable from both the horizontal and the vertical view, though it is only in the vertical view that the user is able to view a graphical display of the borehole.

When selecting the option the **Add new borehole** dialog box appears where the user is prompted to specify the most necessary information about the borehole. Further information could be added by adding it directly to the source database files (the **ADM** and the **LIT** file).



Only the essential information should be given, i.e. all the information in the **Add New Borehole** dialog box is required.

### **10.2.1 Recommended procedure**

- Set the x- and y-co-ordinate, for the new borehole, by typing the co-ordinates, or by clicking the tool button to activate on-screen location of the borehole in the horizontal view. In the vertical view the user has to locate the along profile position by clicking on the view, the connected x- and y-co-ordinates are then automatically calculated.
- Type the borehole ID in the text box.
- Type the ground level in the text box.
- Define the lithology by defining the number of layers in the borehole.
- Type the layer bottom of each layer, the soil type (e.g. sand), and the symbol (e.g. ds).
- In the vertical view use the **Make Graphical Presentation** button to display a graphical presentation of the new borehole.
- Press the **Add Borehole** button to add the borehole to the project.
- The borehole will then be saved to the original database and shown on the view as a green mark, user-defined borehole.



Figure 29 Add New Borehole Dialog Box.

### 10.2.2 Content of add new borehole dialog box

**Set Co-ordinates:** The x- and the y-co-ordinate for the new borehole can be typed directly in the two textboxes or defined by pressing the point-tool in the dialog box followed up by clicking on the borehole location in the horizontal view. In the vertical view the user will have to mark the along profile position, and then get the x- and y-co-ordinates calculated.

**Borehole ID:** Type the ID code for the new borehole, e.g. 214.483.

**Ground Level [m]:** Type the ground level for the new borehole in meters above sea level.

**Number of Layers:** Type the number of geological layers in the new borehole. In the list box below the amount of records corresponding to the number of layers will be added.

The list box is subdivided in four columns:

- 1) Layer No.: The number of the layer. This is the only column that will be filled in automatically.



- 2) Layer Bottom: The user should define the bottom of the layer in meters from the top of the borehole.
- 3) Soil Type: a short description of the geological layer e.g. sand wet or clay.
- 4) Symbol: The geological code for the layer, the geological code should correspond to the codes defined in the ASCII file, see Appendix 1.

**Add Borehole:** To add the new borehole to the project press the **Add Borehole** button.

The co-ordinates, the Borehole ID and the ground level will be saved in the administrative table (**ADM** file), the co-ordinates, the layer bottoms, soil types and soil symbols will be saved in the lithology table (**LIT** file).

### 10.3 Add Borehole Database

This option enables the user to include extra boreholes in the present project. The option is useful if a model area should be extended or if the user missed some boreholes in the initial selection.

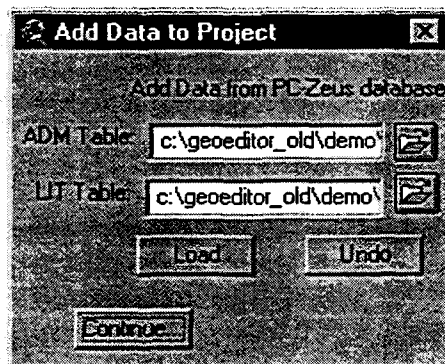


Figure 30 Add Data Dialog Box.

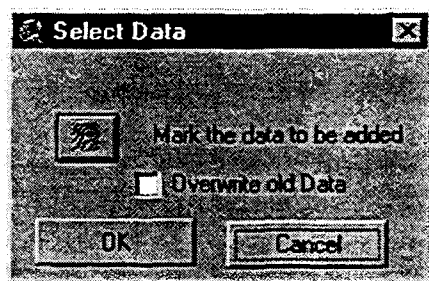


Figure 31 Select Data Dialog Box.



### 10.3.1 Recommended procedure

- Define the source files, i.e. the **ADM** and the **LIT** Table, from which the new data should be extracted.
- Press the **Load** button. The boreholes in the source files are added to the project and presented in a new temporary theme called **added\_data**.
- Press **Undo** to undo the selection.
- Press **Continue** to open the **Select Data** dialog.
- Select the data to import using the **Mark the data to be added** tool. Select the data by drawing a polygon on the screen. Double click to end the polygon.
- All the boreholes inside the polygon will change colour to indicate that they have been selected.
- Change the selection by drawing a new polygon.
- If the any doublets in the imported data should overwrite the old data, click the **overwrite old data** on.
- Press the **OK** button to add the data to the project, or the **Cancel** button to cancel the procedure.

If the **Import** button is pressed, the selected data will be added to the project (inserted in the Boreholes theme), and marked as added data (blue mark).. The temporary **added\_data** theme is deleted automatically.

### 10.3.2 Content of the add data dialog box

**ADM Table:** Specify the ADM table to add. See Appendix 2 for further description.

**LIT Table:** Specify the LIT table to add. See Appendix 2 for further description.

**Load:** The boreholes in the source files are added to the project and presented in a new temporary theme called **added\_data**.

**Undo:** Undo the selection, removes the **added\_data** theme from the view.

**Continue:** Opens the **Select Data** dialog box.

### 10.3.3 Content of the select data dialog box

**Mark the data to be added:** Select the data to import by drawing a polygon on the screen. Double click to end the polygon.





**Overwrite old data:** When the new boreholes are imported a check for borehole duplicates are done. If the **overwrite old data** is checked on the old data is replaced with the imported data.

**OK:** Extracts the selected data, and merges the old borehole database with the selected data.

**Cancel:** Cancel the import procedure.

## 10.4 Borehole Selection

Opens the borehole selection dialog, see Section 5.5.

## 10.5 Add Geophysical Data

TEM data and geoelectrical data (with a Wenner configuration) can be imported.

The data are imported to the horizontal view as a point theme, independent of the geological approach, from here the data can be used in further analysis.

When selecting the **Load geophysical data** from the main menu, the **Add Geophysical Data** dialog box will appear. Pressing the **Add** button for one of the data types opens a browse box where the user should locate the source file.

For a more detailed description of the file format see Appendix 2.

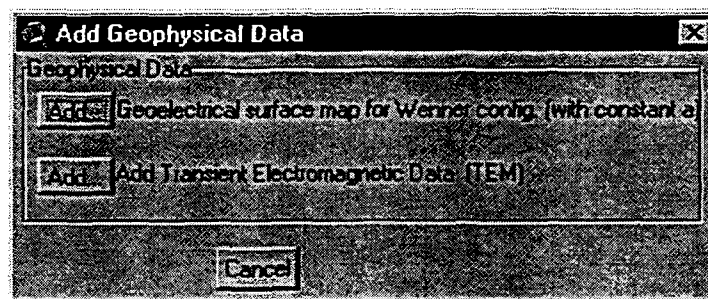


Figure 32 Add Geophysical Data Dialog Box.



## 11 BOREHOLE INFORMATION AND EDITING TOOL

The borehole information and editing tool is accessible as a tool button in the view and as a tool in the tools, see Chapter 13.



Figure 33 Borehole Information and Editing Tool.

The borehole information and editing tool enables the user to view the basic data for the selected borehole and a profile showing the geology with marked water level and screen positions. The borehole information tool is enabled for all the views in the GeoEditor.

### 11.1 Recommended Procedure

- Select the tool by pressing the tool button located in the toolbox or in the view.
- Select a borehole by clicking on the borehole either in the horizontal view, in the vertical view or in a close-up view. View the borehole information by toggling between the different tabs.
- The Info tab displays the basic borehole information; co-ordinates, ground level etc.
- The dimensions tab displays the dimensions of the borehole; dimensions for the borehole, the casing and the screen.
- The Hydr. Data tab displays all the hydraulic data associated with the borehole.
- Pressing the **Geology** button will open a view containing a geological profile including the screen and water level positions. From this view the user is able to change the geology, the screen position or the water level position and save the new information to the database.



**Borehole Information**

Info | Dimensions | Hydr. Data

Borehole ID: 98. 741

Date: 19800125

Case No.:

Area Code: 737

Drilling Place: Illerup Vandværk, Krogdalsevej

Driller:

X Coord: 554368

Y Coord: 6213400

Elevation: 62

Purpose: Vandforsyningsboring

Method: Rotaryboring

User:

Close Geology

Figure 34 Borehole Information Dialog Box.

## 11.2 Geology

When pressing the **Geology** button in the **Borehole properties** dialog box the lithology, the screen position and the water level measurements of the selected borehole are extracted from the database and shown in a view, see Figure 35.

The **Display Borehole** dialog box enabling the user to change the parameters of the borehole accompanies this view, see Figure 35 and Figure 36.

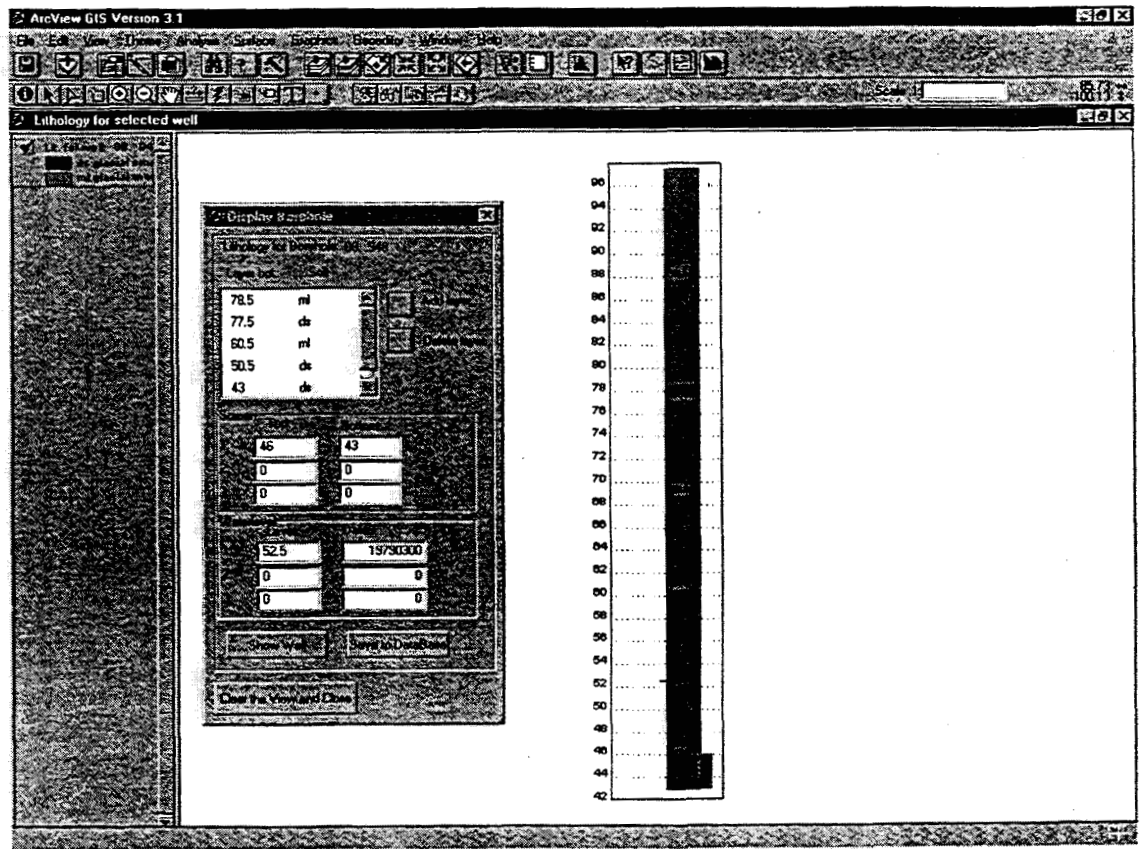


Figure 35 View of the Lithology of the Selected Borehole.

### 11.2.1 Contents of the display borehole dialog box

**Lithology of the Borehole:** Displays the lithology of the borehole by layer bottom and soil ID. The table is edited by selecting a cell followed by typing a new value.

**Add layer:** Adds a new layer to the lithology list. The parameters of the layer are entered by selecting a cell and typing the new value.

**Delete layer:** Deletes the selected layer.

**Screen:** Displays the screen information of the selected borehole. The screen information is also showed graphically in the view. The screen information is edited by typing new values in the textboxes.

**Water level:** Displays the measured groundwater level position and the date of measurement for the borehole. The water level information is also displayed graphically in the view. The water level information is edited by typing new values in the textboxes.



**Show Well:** Shows a graphical display of the borehole with the parameters defined in the dialog box, without saving the parameters to the database. This feature is useful when viewing changes that are not to be saved.

**Display Borehole**

Borehole ID Borehole: 88-948

Layer bol      Sp

78.5	ml
77.5	ds
60.5	ml
50.5	ds
43	ds

Depth

	Top	Bottom
1	46	43
2	0	0
3	0	0

Waterlevel

	Level	Date
1	52.5	19790300
2	0	0
3	0	0

Show Well      Save to Database

Clear the View and Close

Figure 36      *Display Borehole Dialog Box.*

**Save to Database:** Extracts the parameters from the dialog box and saves them to the database. Any predefined parameters of the database are overwritten.

**Clear the View and Close:** Deletes the view and reopens the Borehole properties dialog box.



## 12 SETTINGS

The settings dialog is assessable through the GeoEditor menu, and through right-mouse click in both the horizontal and vertical view. The settings dialog contains all the settings for the GeoEditor project. The settings are divided in four groups; Axis, Show data, GeoEditor and Misc.

### 12.1 Axis Settings

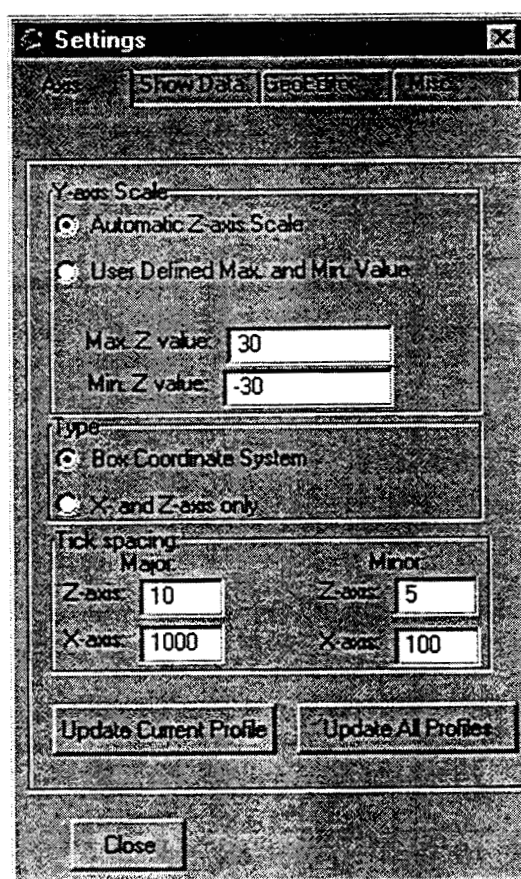


Figure 37 The Axis Settings.

The axis setting group are used to define the co-ordinate system in the vertical view. The axis settings are used every time a new vertical view is created.

**Automatic Z-axis scale:** When creating a vertical view the z-axis is scaled after the minimum and maximum values in the view.



**User Defined max. and min. value:** When creating a vertical view the z-axis is scaled after user defined values. The maximum and minimum z-values are typed in the two textboxes.

**Box co-ordinate system:** Display the co-ordinate system as a box-co-ordinate system, left and right z-axis, and top and bottom x-axis.

**x- and z-axis only:** Display the co-ordinate system with only one z-axis and only one x-axis.

**Tick spacing:** Specify the tick spacing for the co-ordinate system.

**Update current profile:** Update the current vertical view after the axis settings.

**Update all profiles:** Update all the vertical views after the axis settings.

## 12.2 Show Data Settings

The show data setting group are used to define which data to be displayed in the vertical view. These settings are used every time a new vertical profile is created.

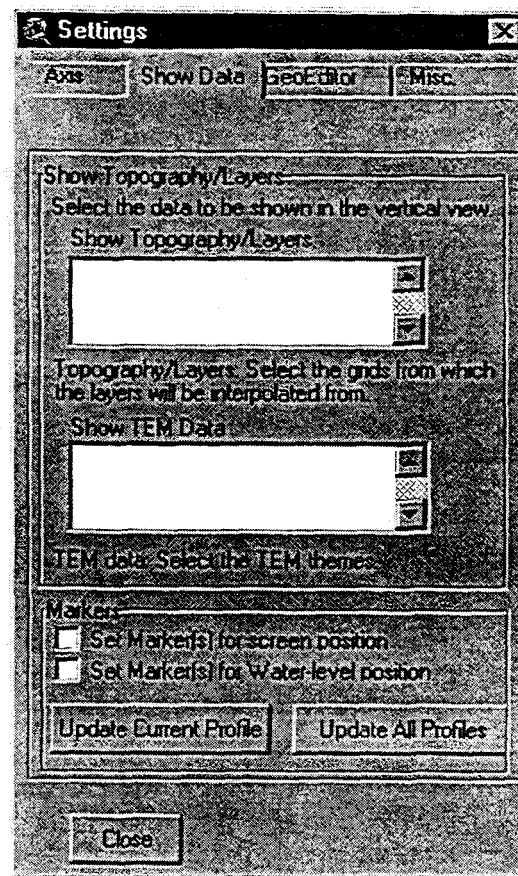


Figure 38 Show Data Settings.

**Show Topography/Layers:** The list box displays all the grid themes in the horizontal view. When selecting themes from the list, a layer from each grid-theme will be added to all the created 2D-view. Grid values along the profile line will be extracted and displayed in the 2D-view as a polyline-theme. If a layer should be imported to an already generated vertical view, use the **Add layer or TEM data to view** tool in the tools, see Chapter 13.

This could be used to display topography or interpolated layers.

**Show TEM data:** TEM data can be added to the vertical view for additional geological information. Prior to the addition the TEM data must be loaded in the horizontal view, see Section 10.5. Thus the TEM data will appear in the list box. To add the data, select the appropriate grid themes in the list box. Then they will be added to all the vertical-views created.

Notice that only the TEM data with a depth positioned within the interval for the y-axis in the vertical view will be selected. A point theme containing the location and the resistivity is created. The point theme will adopt the geological codes defined in the resistivity filter. Use the **Resistivity Filter** option to set the filter, see Section 12.4.





**Set markers for screen position:** Set markers for screen position or water level. Use the update button to update already created vertical views to new marker settings.

**Set markers for water-level position:** Set markers for water-level position. Use the update button to update already created vertical views to new marker settings.

**Update current profile:** Update the current vertical view to the new marker settings.

**Update all profiles:** Update all the vertical views to the new marker settings.

### 12.3 *GeoEditor Settings*

The GeoEditor settings are used to define the GeoEditor folder and the model approach. These settings should already be defined, as this is a requirement for starting a GeoEditor project.

**Set GeoEditor Folder:** Set the GeoEditor project folder, which is used to save all the files, created during a GeoEditor project.

**Set Model Approach:** Specify the model approach to use. **Vertical profiles:** Develop a geological model, consisting of geological layers interpolated from discrete points. Define horizontal profiles and digitise discrete points representing a selected lithology for each profile. Interpolate geological layers from the digitised points and export the layers to an external format. **Depth intervals:** Develop surface maps representing the dominant lithology in specified layers. Classify the lithology, specify depth intervals for subtracting the dominant lithology, mark areas or define by Thiessen polygons and export to external format.

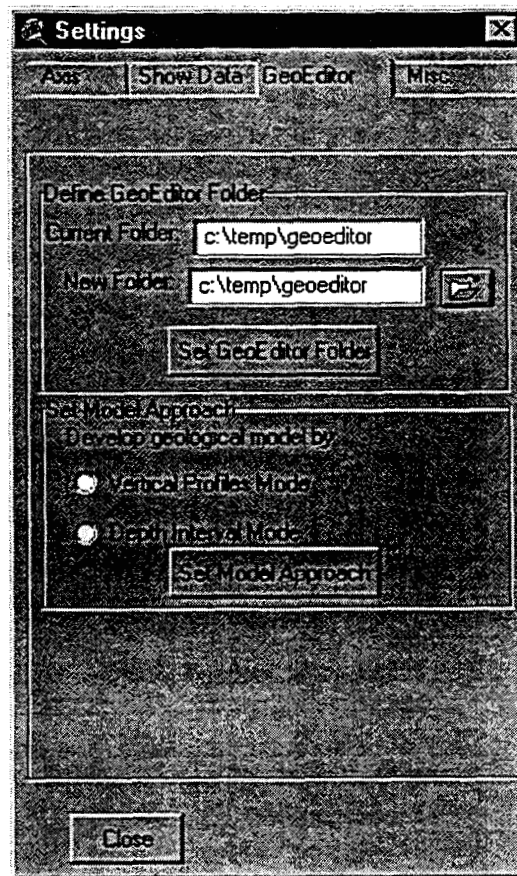


Figure 39 GeoEditor Settings.

## 12.4 Misc. Settings

Miscellaneous settings contain different settings for the GeoEditor .

**Cascade 2D-views:** This option will cascade the vertical views.

**Always update vertical views:** When this option is checked on the vertical view are always created when opening. If this option is not checked on already created views are not created again.

**Overlapping boreholes:** When boreholes overlap in the vertical view there are two options for presenting the profiles. **Show all with true position** is the default option and will show the boreholes at their true position. If several boreholes overlap they will be drawn on top of each other. **Show all with adjusted position:** Shows the boreholes so that all the boreholes will be visible. If several boreholes overlap the positions will be adjusted so that they are all visible. This option is not recommended since it gives an illusionary impression of the geology.



**Set language:** Shows the field names in the selected language. All field names of the Boreholes theme (administrative and lithological data) are shown in the selected language. By default the field names are given in Danish but can optionally be translated to English.

## 12.5 Resistivity Filter

This submenu activates a dialog box showing the default resistivity filters. This is used to convert resistivity values, in ohmm, to geological symbols.

### **Connection between resistivity and geological layer.**

The dialog box is divided into three columns:

- 1) Resistivity from and resistivity to: these two columns define a resistivity interval, in ohmm.
- 2) Description; a short description of the geological layer e.g. sand wet or clay.
- 3) Code; the geological code for the layer. This code will be used to identify the layer.

When the dialog box appears it will always contain some values. These values are read from the resistivity.dbf file in the \$Geoeditor\Data directory. This file will either contain the default values or the values from the last modification.

The dialog box enables the user to change, add or delete values/records.

**Change Values:** To change some already defined values, press the **Change Values** radio button, and then the OK button. The **Change Values** box will then be enabled, and the user should;

- 1) Select the cell containing the value to be changed.
- 2) Type the new value in the textbox.
- 3) Press the OK button. The new value will be saved to the resistivity.dbf file.

**Add Values:** To add new values press the **Add Values** radio button, and then the OK button. An empty record will be added at the bottom of the list. To add values to the new record, follow the procedure for changing values.

**Delete Record:** To delete records press the **Delete Record** radio button, and then the OK button. The **Delete Values** will be enabled, and the user should;

- 1) Select the row to be deleted.
- 2) Press the OK button. T

The selected row will be deleted from the list and from the resistivity.dbf file.



**Important notice:** As the dialog box is directly linked to the resistivity.dbf file and all changes are saved immediately it is not possible to undo changes. If the user wants to save a special resistivity set-up, please make a copy of the resistivity.dbf file and rename it. To activate an old set-up, rename the old set-up file by:  
\$Geoeditor\Data\resistivity.dbf.

## 12.6 Lithology Colours

Creates a new view containing the colour scheme for the different geological formations.

**Resistivity Values**

Resistivity (ohmm)		Description	Code
From	To		
1	10	Fed ler	l
11	40	Moræne ler	ml
41	99	Moræne sand	ms
101	200	Sand u. grund. spejl	ds
201	10000	Sand tør	fg

Change/Add/Delete Values  
☒ Change Values ☐ Add Values ☐ Delete Record

Change Values

1) Select cell

2) Type new Value:

3) Press button:

Delete Values

1) Selection

2) Press button:

Figure 40 The Resistivity Filter Dialog Box.





## 13 TOOLS

All the utility tools associated with the GeoEditor are located in the tool dialog box, accessible through the tools menu in the GeoEditor main menu.

The tools are subdivided in to tools used when the horizontal view is active, and tools used when the vertical view is active. The tools are only enabled according to their respective active view.

### 13.1 Tools for the Horizontal View

#### Borehole selection



Re-opens the Borehole selection dialog box. This tool only works when the horizontal view is active.

#### Borehole information



This tool is used to identify the parameters of a selected borehole. The tool will only work when the borehole theme is active. To use the tool press the tool button and click on a borehole. A dialog box containing all the basis information about this borehole will appear. Three sub-dialog boxes are available by clicking on one of the buttons;

**Properties:** prompts a dialog box showing the borehole properties.

**Test pump:** prompts a dialog box showing the test pumping data for the borehole (if any). **Geology:** opens a new view containing a profile of the lithology in the selected borehole, enabling the user to change the geology or the screen positions.

See Chapter 11 for further description.

#### Select Profile



If the horizontal view is active this tool will show the 2D-view for a selected profile. Activate the tool, by clicking, click on the profile to select and the associated 2D-view will appear.

320



## 13.2 Tools for the Vertical View

### Add Layer or TEM data to view



This button only works when the 2D-view is active. Adds a layer from a grid-theme to the 2D-view. When pressing the "Add Layer or TEM data button" a dialog box appear with a list showing all the grid themes in the horizontal view. To select a grid-theme select the theme from the list and press the "Add to view" button. Then grid values along the profile line will be extracted and displayed in the 2D-view as a polyline-theme.

This tool is useful in the attempt to display topography, or interpreted layers. If TEM data is imported to the horizontal view they could also be imported to the vertical view using this tool.

### Cascade 2D Views



If the cascade option is clicked on the 2D-views will appear in cascade.

### Tile 2D Views



Tiles all the vertical views.

### Select tiled view



Select tiled view.

## 13.3 Extract Data Tools

### Water Level

Extracts the water level for all the boreholes in the borehole theme. The output will be an interpolated surface map. Notice the water levels in the database file could have been measured at different times. As the water table in the model area usually fluctuates throughout the year the extracted water levels could be misleading.

### Layer thickness

Calculates the layer thickness. The ability requires two active layer surfaces: one representing the upper level and another representing the lower level. The output is a surface map. This tool can sequentially extract the thickness of each layer.



### UZ/SZ

Extract the unsaturated and the saturated zones. This ability requires an active water level surface map and an active layer surface map.

### Transmissivity

Extract the transmissivity for all the boreholes as a surface map.

### Filter position

Extracts boreholes with filter in selected layer. Requires two active layer surfaces. The output is a point theme representing the boreholes with filter in the selected layer.

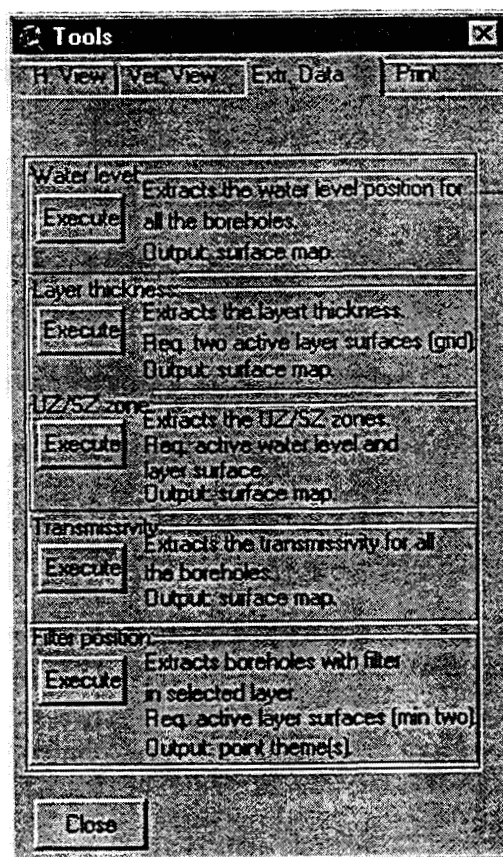


Figure 41 Extract Data Dialog Box.





## 13.4 *Print Profiles*

The **Print profiles** option, enables the user to make a standard layout containing the defined profiles. In the present version the layout is only generated but not printed, so the user will have to open the layouts, in the ArcView® menu: View/Layout and print them manually from there.

### 13.4.1 *Recommended procedure*

- Select the profiles to export from the list. Make a multiple selection by holding the shift button down while selecting.
- Select the Export button to make the layouts. The layouts will be called layout + the profile name.
- Open the layouts, for modification and print.

### 13.4.2 *The content of the export profile dialog box*

**Select the views to be exported:** Contains all the defined vertical views. Select the views to be exported.

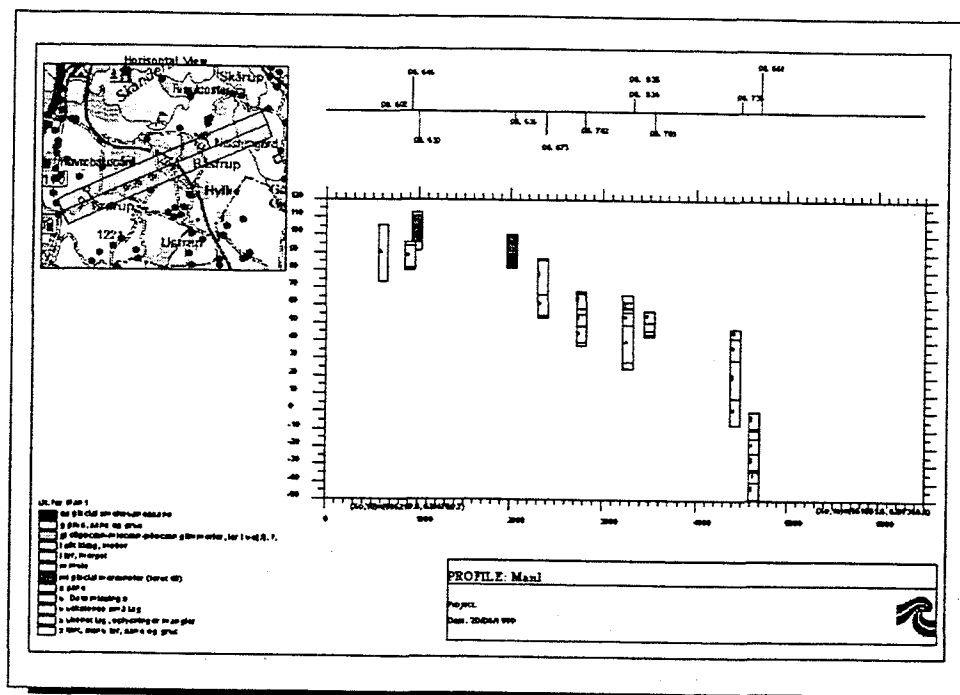
**Select All:** Selects all the views in the list.

**Export to printer:** Exports the selected views to the ArcView® layout format.

**Printer Set up:** Opens the printer set up dialog box.

**Project Description:** A description typed here will be added to all the layouts.

**Export:** Generates the layouts, see Figure 42.



324





## 14 FILE CONVERSION

The File Conversion option enables the user to convert a borehole database format to the PC-Zeus format supported by the GeoEditor. The file conversion is accessible from the project window in the GeoEditor menu.

When selecting the File Conversion menu an extension containing the file conversion is loaded. The file conversion requires the source format to be importable by ArcView. Thus two optional file types can be imported: dBase format and tabulator or space delimited ASCII format.

### 14.1 Recommended Procedure

- Select the **File Conversion** in the GeoEditor menu in the project window. The **Convert Table** dialog box will open, see Figure 43.
- Import the files containing the borehole information. The table files should contain administrative and lithological information. One or more files could hold the information.
- Choose language.
- Press **Define Fields** to proceed. This will open the **Define Fields** dialog box.
- For each item supported by the data: select the table and the field in the table containing the data. Data corresponding to the items has to be selected
- Press **Screen Data** to define any screen data specified in the same way as the administrative data.
- Press **Lit Data** to define the lithological data specified in the same way as the administrative data.
- Press **Back to main** to return to the **Convert Table** dialog box.
- Specify the output directory for the output files (ADM and LIT file).
- Press **Make Table** to create data files in PC ZEUS format. The output files will be saved in the specified output directory.
- The new set of administrative and lithological data can be incorporated in a new project or in an existing project by using the **add data to project** function from the GeoEditor main menu category Miscellaneous.



## 14.2 Content of the Convert Table Dialog Box

**Select the files to import:** Opens a browse dialog box enabling the user to import dbf or ASCII files. The files are imported by ArcView as tables. When importing files the GeoEditor checks for multiple files and empty files.

**Imported tables:** Shows a list of the imported files.

**Define Fields:** Opens the Define Fields dialog box, from where the user is able to specify the links between the tables and the fields.

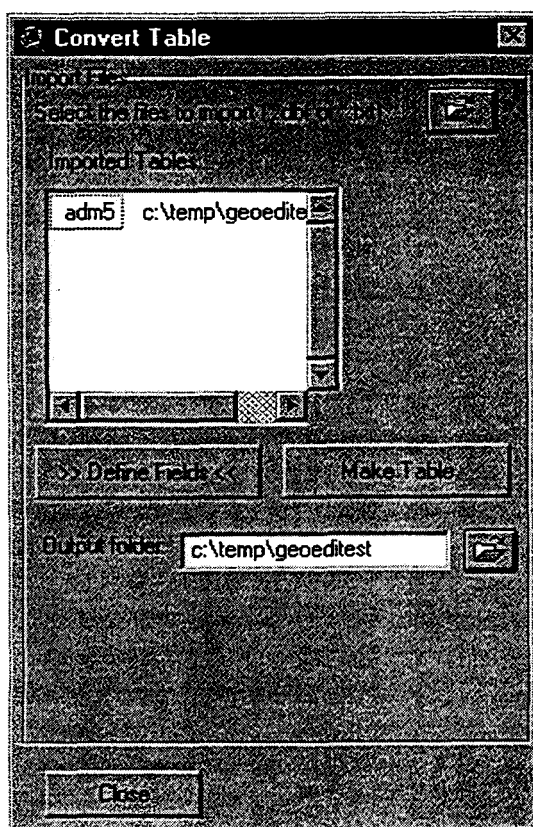


Figure 43 Convert Table Dialog Box.

**Make Table:** Starts the generation of the PC-Zeus files employing the user specified parameters.

**Output folder:** Specifies the folder for the output files. If the administrative as well as the lithological data are specified, then an ADM and a LIT file will be generated.

**Close:** Closes the dialog box and uninstall the File Converter extension.



### 14.3 Content of the Define Fields Dialog Box

The **Define Fields** dialog box is accessible from the **Convert Table** dialog box, and enables the user to specify the links between the imported tables and the needed fields.

**Field:** The fields listed are the available fields for the administrative part of the database.

**Table:** For each of the fields, or as many as the source data supports, specify the imported table of data.

**Field:** Specify the field in the specified table containing the data.

**ID Field:** If the specified table is different from the table containing the ID's the user will have to specify which field contains the ID's. This is needed as the tables are merged based on a common ID.

**Back to main:** Reopens the **Convert Table** dialog box.

**Lit. Data:** Opens the **Lithological part** dialog box.

**Screen Data:** Opens the **Define Screen Data** dialog box.



Field	Table	Field	ID Field
ID	adm5	Dgunr	
Location	None		
X-coordinate	adm5	Utmx	
Y-coordinate	adm5	Utmv	
Municipality code	None		
Surface Level	None		
Construction date	None		
Type	None		
Depth to Wastewater	None		
Date of Measurement	None		

<< Back to Main   Exit Data >>   Screen Data >>

Figure 44 Define Fields Dialog Box.

#### 14.4 Content of the Define Screen Data Dialog Box

The **Define Screen Data** dialog box enables the user to specify screen data.

**Field:** The fields listed are the available fields for the administrative part of the database.

**Table:** For each of the fields, or as many as the source data supports, specify the imported table where the data could be found.

**Field:** Specify the field in the specified table containing the data.

**ID Field:** If the specified table is different from the table containing the ID's the user will have to specify which field contains the ID's. This is needed as the tables are merged based on a common ID.

**Back:** Closes the dialog box and reopens the **Define Fields** dialog box.

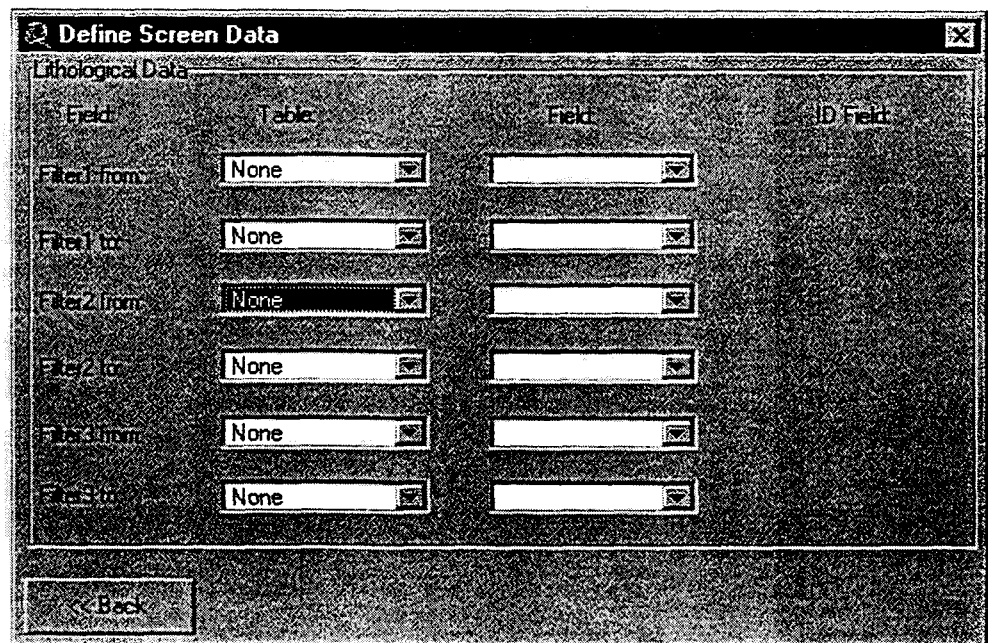


Figure 45 Define Screen Data Dialog Box.

### Content of the Lithological Part Dialog Box

The **Lithological part** dialog box enables the user to specify the lithological part of the database.

**Field:** The fields listed are the available fields for the administrative part of the database.

**Table:** For each of the fields, or as many as the source data supports, specify the imported table where the data could be found.

**Field:** Specify the field in the specified table containing the data.

**ID Field:** If the specified table is different from the table containing the ID's the user will have to specify which field contains the ID's. This is needed as the tables are merged based on a common ID.





**Back:** Closes the dialog box and reopens the **Define Fields** dialog box.

Field	Table	Field	ID Field
ID	None		
Layer bottom	None		
Ecological Symbol	None		

Back



## 15 TUTORIAL AND HELP SYSTEM

Two help approaches have been applied in order to make the GeoEditor more user friendly and easier to use:

- On-line help, which basically is the manual in an on-line format
- Tutorial movies describing the basic facilities of the GeoEditor

### 15.1 Online Help

The help system is presently only available from the main menu where the help is invoked by selecting the Help option.

### 15.2 Tutorial Movies

The Tutorial movies are accessible from the **GeoEditor** menu in the Project window. When selecting the **Tutorial** option the Tutorial dialog box appears.

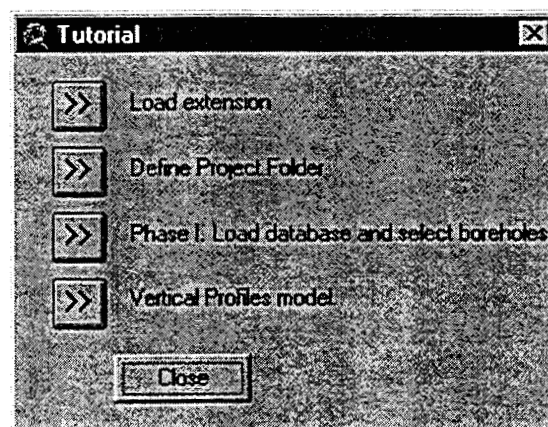


Figure 46 Tutorial Dialog Box.

#### 15.2.1 Recommended procedure

- Select the desired tutorial to run.
- If the user wants to play several movies in a row the Screen Cam program has to be closed before another execution (simply close the small play control window).





## APPENDIX





## Appendix 1

### Description of the files in the GeoEditor folder

The GeoEditor folder is used by the GeoEditor to access necessary files. The content of the files in each of the subfolders will be explained below. The GeoEditor folder contains four subfolders: GeoEditorFiles, Help, Tutorials and Examples. The GeoEditorFiles folder then contains two additional subfolders Legends and Maps.

#### The GeoEditorFiles Folder

This folder contains the files used by the GeoEditor, to convert codes to a description and executable subprograms called by the GeoEditor.

A short description of the files in the SystemFiles folder.

**Toolsdll.dll:** This dll is called from the GeoEditor when generating the project folder and its subfolders.

**Resistivity.dbf:** This file is a database file containing the default connection between resistivity [ohmm] and a geological description.

Table 1 *Resistivity.dbf file*

RESA	RESB	DESCR	CODE
1	10	Fed ler	l
11	40	Moræne ler	ml
41	99	Moræne sand	ms
101	200	Sand u. grund. spejl	ds
201	10000	Sand tør	fg

The fields used are:

RESA: The start value of the resistivity interval.

RESB: The end value of the resistivity interval.

DESCR: The geological description.

CODE: The lithology code, this code should be included in the zeuskoder.txt file.

**Anv.dbf:** This file is a database file containing the default connection between the codes describing the use of the borehole and the associated description.

The fields used are:



Table 2 Anv.dbf file

CODE	USE
A	Andet
D	Dybhulsproduktion
I	Vandinjektion
M	Moniteringsboring
N	Ingen anvendelse
O	Opgivet boring
P	Pejlestation
S	Sløjfet boring
V	Vandindvinding

CODE: The code describing the use of the borehole.

USE: The description of the code.

**Metode.dbf:** This file is a database file containing the default connection between the codes describing the method used for constructing the borehole and the associated description.

The fields used are:

CODE: The code describing the method used for constructing the borehole.

USE: The description of the code.

Table 3 Metode.dbf file

CODE	USE
A	Andet
B	Botesam/rammeboring
D	Direkte skylleboring
E	El-log boring
G	Snegleboring
I	Indirekte skylleboring
L	Luftskylleboring
P	Pneumatisk boring
R	Rotaryboring
S	Skyleboring
T	Tørboring/slagboring
V	Sugeboring

**Formaal.dbf:** This file is a database file containing the default connection between the codes describing the purpose of the borehole and the associated description.



Table 4      *Formaal.dbf file*

CODE	USE
A	Andet
B	Brunkulsboring
C	Brønd
D	Dybdeboring
F	Frederikshavn gasboring
G	Geoteknisk boring
I	Videnskabelig boring
L	Miljøundersøgelser
M	Moniteringsboring
P	Pejleboring
R	Råstofs boring
S	Shot hole
U	Prøveboring
V	Vandforsyningsboring

The fields used are:

CODE: The code describing the purpose of the borehole.

USE: The description of the code.

**Zeuskoder.txt:** This file is an ASCII file containing the link between the code describing the lithology and the associated description and a colour code for each description.

Table 5      *Zeuskoder.txt file*

fg	postglacial ferskvandsgrus	50	254	0
fp	postglacial ferskvandsgytje	128	152	26
fl	postglacial ferskvandsler	179	229	0
fs	postglacial ferskvandssand	131	229	0
fi	postglacial ferskvandssilt	153	229	0
ft	postglacial ferskvandstørv	77	152	26
fv	postglacial vekslende små ferskvandslag	2	229	0
es	postglacial flyvesand	253	254	25
fk	postglacial kildekalk	2	178	0
fj	postglacial okker	1	127	0
hg	postglacial saltvandsgrus	76	204	254
hs	postglacial saltvandssand	72	95	100
hi	postglacial saltvandssilt	32	100	100
ht	postglacial saltvandstørv	32	100	100

The ASCII file has to comply by the format shown above;

The first column has to contain the codes, as a string or as a number.

The last three columns have to contain the colour codes in RGB

values. The string in-between the first and the third last columns are





translated as the description of the lithology. The description could be in any format, e.g. containing blanks or numeric characters. The four columns are tab-separated.

### **The Legends Folder**

This folder contains the legend files, \*.avl files, used by the GeoEditor to set some standardised legends for some themes. If the user wants to change the appearance of some themes a new avl files can be made. Open the legend editor, modify the present legend and save the legend as the standard avl file replacing the old avl file.

A short description of the files in the legends folder and their associated themes:

Modellegend.avl : Sets the legend for the Boreholes theme.

Templegend.avl : Sets the legend for imported TEM data.

Profilelegend.avl : Sets the legend for the Profiles theme.

Firstlegend.avl : Sets the legend for the boreholes in the first view.

### **The Maps Folder**

This folder contains the standard maps used by the GeoEditor. Only a standard map for the first view is present.

A short description of the files in the maps folder.

Background.bmp: Bitmap of Denmark in 1:2000000 scale. This bitmap is used in the first view, as a base-map for the initial selection. If the user wants another map as default bitmap it should be saved with this filename.

Background.bpw: This file contains information about the georeference of the dk.bmp file. Consult the section of "worldfiles" in the ArcView® help menu for more information.

### **The Help Folder**

The Help folder contains the files used by the online help system.



### **The Tutorial Folder**

The Tutorial folder contains the files used when running the tutorial movies.

**Scplayer.exe:** Freeware program used to play the Screen Cam 97 movies when running a Windows machine.

**Scplaynt.exe:** Freeware program used to play the Screen Cam 97 movies when running a NT machine.

### **The Examples Folder**

The examples folder contains two dbf files in order to run the examples shown in Appendix 3.

337





## Appendix 2

### File Format

#### T2 FILE

The T2 file is an ASCII file used for matrix data by MIKE SHE.

```
FILETYPE DATATYPE Verno: 22 100 502
NX NY DIM Xorig Yorig : 110 110 5.0000000E+001 -2.8347200E+005 5.2711996E+004
DELETE UTMZONE ORIENT : -1E-035 0 0.000000
MIN MAX MEAN ST.DEV : 1.803138E+001 2.809112E+001 2.123936E+001 1.703041E+000
Theme: grid3
110
20.0172 20.04639 20.03877 20.03072 20.02321
20.01772 20.01595 20.0356 20.05353 20.08295
20.12196 20.16773 20.25547 20.31303 20.45654
20.51945 20.57864 20.7403 20.79562 20.84761
```

Figure 47 The Format of the T2 File

The header lines (5) specifies which kind of data is in the file. Each line in the header consists of 24 characters of text (which will be skipped when reading) followed by information;

Line 1: Specifies the data file type, the data type and the version number (530 for MIKE SHE version 5.3).

File type: 21- the data in the data file is considered to be integers (grid codes) even though the data is written as reals.

22-the data in the data file is considered to be reals even though the data is written as integers.

Data type: the data type is presently not used but the only allowable type is:  
57: Any grid data.

Line 2: Specifies the dimensions and origin of the data;

nx, ny: dimensions of the matrix.

dim: dimension of the grid squares.

xorig, yorig: location of the origin in the co-ordinate system.



Line 3: Defines a delete value that defines areas with missing values or outside the catchment, the UTM zone (specify a zero if unused) and the orientation of the matrix data.

Line 4: Defines some statistical parameters for the data values in the data file.

Line 5: text line.

### **Grid Definition File**

The grid definition file is an ASCII file used to save a grid set-up. The format of the grid definition file is used for all the MIKE SHE – GIS extensions with respect to grid definition.

```
Grid Properties for MikeShe GIS
Grid properties Tinglev Mose 50 meter
xorg. -283471.9981
yorg. 52711.9960
Columns 110
Rows 110
Cellsize 50
```

*Figure 48 The Grid Definition File.*

Line 1: This line identifies the file as a grid definition file.

Line 2: User comments.

Line 3: The x-co-ordinate for the lower left corner.

Line 4: The y-co-ordinate for the lower left corner.

Line 5: The number of columns.

Line 6: The number of rows.

Line 7: The dimension of the grid cells (squares).



## Geophysical Data Files

The user can import TEM and geoelectrical data.

### TEM Data

The following format is used for TEM data.

UTMX	UTMY	40 V	40 H	30 V	30 H	20 V	20 H	10 V	10 H	0 V	0 H	-10 V	-10 H
584824.3	6127835.0	12.488	9.040	60.106	32.468	77.765	75.532	85.900	85.900	85.900	85.900	85.900	85.900
584996.0	6127750.0	27.330	27.330	63.162	38.965	100.539	70.091	89.873	69.772	50.798	45.222	41.850	
584460.8	6127737.0	9.240	9.240	97.571	35.012	92.183	68.691	60.559	55.988	51.990	51.990	51.990	
584365.8	6127528.0	27.578	8.773	104.596	38.548	90.566	80.311	60.346	59.953	59.560	59.560	59.560	
584205.6	6127364.0	-9999	-9999	51.562	30.381	85.781	48.486	94.055	66.316	50.080	37.717	32.050	
584025.4	6127159.0	-9999	-9999	58.055	29.740	89.028	47.666	82.303	57.020	39.368	35.688	34.130	
584044.1	6126934.0	21.110	21.110	45.609	31.740	63.054	49.484	73.150	73.150	73.150	73.150	71.376	

The first line is a text line, the rest is data lines.

Column 1: The x-co-ordinates.

Column 2: The y-co-ordinates

Column 3-n: The resistivity values [ohmm] in the different depths and in the horizontal and vertical direction. The GeoEditor only uses the vertical data.

### PC Zeus Files

The GeoEditor has been made to use the PC Zeus database files as input files.

This format is developed by the Geological Survey of Denmark and Greenland (GEUS).

The data are subdivided in two file:

One describing the administrative data, the **ADM** file.

One describing the lithology, the **LIT** file.

The **ADM** and the **LIT** files are both employing the dBase format.

### The ADM File

An example illustrating the different features of the **ADM** file..

The names of the features are given in Danish (Name (DK)) but in the table in addition an English name and description is included.

The column "Type" describes the format; for strings the length and a 'c', for numeric the length a dot the number of decimals and a 'n'.



Name (DK)	Name (UK)	Description	Type	Used by the GeoEditor
DGUNR	ID	BOREHOLE ID	10c	x
BB	DRILLER	Borehole DRILLER	18c	x
KOM	MUNICIPALITY	MUNICIPALITY	3n	x
STED1	PLACE1	PLACE 1	60c	x
STED2	PLACE2	PLACE 2	60c	
SAGSNR	ACCNO	ACOUNT NUMBER	13c	
LOEBNR	SERNO	SERIAL NUMBER	13c	
UTMX	XCOR	X-COORDINATE	6n	x
UTMY	YCOR	Y-COORDINATE	7n	x
KOTE	LEVEL	GROUND LEVEL	6.2n	x
DATO	DATE	DATE	8n	x
FORMAAL	PURPOSE	PURPOSE OF BOREHOLE	1c	x
METODE	METHOD	DRILLING METHOD	2c	x
ANV	USE	BOREHOLE USE	2c	x
BORDIA1	WELLDIA1	BOREHOLE DIAMETER 1	4n	
BORTCM1	WELLUNIT1	UNIT FOR ABOVE	1c	
BORDYB1	WELLDEPTH1	DEPTH OF PIPE 1	5.1n	
BORDIA2	WELLDIA2	BOREHOLE DIAMETER 2	4n	
BORTCM2	WELLUNIT2	UNIT FOR ABOVE	1c	
BORDYB2	WELLDEPTH2	DEPTH OF PIPE 2	5.1n	
BORDIA3	WELLDIA3	BOREHOLE DIAMETER 3	4n	
BORTCM3	WELLUNIT3	UNIT FOR ABOVE	1c	
BORDYB3	WELLDEPTH3	DEPTH OF PIPE 3	5.1n	
FORDIA1	CASINGDIA1	CASING DIAMETER 1	4n	
FORTCM1	CASINGUNIT1	UNIT FOR ABOVE	1c	
FORDYB1	CASINGDEPTH1	DEPTH OF CASING 1	5.1n	x
FORMAT1	CASINGMAT1	MATERIAL FOR CASING 1	1c	x
FORDIA2	CASINGDIA2	CASING DIAMETER 2	4n	x
FORTCM2	CASINGUNIT2	UNIT FOR ABOVE	1c	x
FORDYB2	CASINGDEPTH2	DEPTH OF CASING 2	5.1n	x
FORMAT2	CASINGMAT2	MATERIAL FOR CASING 2	1c	x
FORDIA3	CASINGDIA3	CASING DIAMETER 3	4n	x
FORTCM3	CASINGUNIT3	UNIT FOR ABOVE	1c	x
FORDYB3	CASINGDEPTH3	DEPTH OF CASING 3	5.1n	x
FORMAT3	CASINGMAT3	MATERIAL FOR CASING 3	1c	x
FILDIA1	SCREENDIA1	DIAMETER OF SCREEN 1	4n	
FILTCM1	SCREENUNIT1	UNIT FOR ABOVE	1c	
FILFRA1	SCREENFROM1	TOP OF SCREEN 1	5.1n	x
FILTIL1	SCREENTO1	BOTTOM OF SCREEN 1	5.1n	x
FILMAT1	SCREENMAT1	MATERIAL FOR SCREEN 1	1c	
FILDIA2	SCREENDIA2	DIAMETER OF SCREEN 2	4n	
FILTCM2	SCREENUNIT2	UNIT FOR ABOVE	1c	
FILFRA2	SCREENFROM2	TOP OF SCREEN 2	5.1n	x
FILTIL2	SCREENTO2	BOTTOM OF SCREEN 2	5.1n	x
FILMAT2	SCREENMAT2	MATERIAL FOR SCREEN 2	1c	
FILDIA3	SCREENDIA3	DIAMETER OF SCREEN 3	4n	



FILTCM3	SCREENUNIT3	UNIT FOR ABOVE	1c	
FILFRA3	SCREENFROM3	TOP OF SCREEN 3	5.1n	x
FILTIL3	SCREENTO3	BOTTOM OF SCREEN 3	5.1n	x
FILMAT3	SCREENMAT3	MATERIAL FOR SCREEN 3	1c	
VSPDYB1	WATDEPTH1	DEPTH TO WATER 1	5.1n	x
VSPDATO1	WATDATE1	DATE 1	8n	x
VSPDYB2	WATDEPTH2	DEPTH TO WATER 2	5.1n	x
VSPDATO2	WATDATE2	DATE 2	8n	x
VSPDYB3	WATDEPTH3	DEPTH TO WATER 3	5.1n	x
VSPDATO3	WATDATE3	DATE 3	8n	x
PEJDYB1	LEVEL1	WATERLEVEL 1	5.1n	
PEJDATO1	LEVELDATE1	DATE 1	8n	
PEJDYB2	LEVEL2	WATERLEVEL 2	5.1n	
PEJDATO2	LEVELDATE2	DATE 2	8n	
PEJDYB3	LEVEL3	WATERLEVEL 3	5.1n	
PEJDATO3	LEVELDATE3	DATE 3	8n	
YDEM31	ABSTRAC1	ABSTRACTION 1	5.1n	x
SAENK1	DRAWDOWN1	DRAWDOWN 1	5.1n	
PUMTID1	PUMPTIME1	PUMP TIME 1	6n	
YDEM32	ABSTRAC2	ABSTRACTION 2	5.1n	x
SAENK2	DRAWDOWN2	DRAWDOWN 2	5.1n	
PUMTID2	PUMPTIME2	PUMP TIME 2	6n	
YDEM33	ABSTRAC3	ABSTRACTION 3	5.1n	x
SAENK3	DRAWDOWN3	DRAWDOWN 3	5.1n	
PUMTID3	PUMPTIME3	PUMP TIME 3	6n	
TRANSMIS	TRANS	TRANSMISSIVITY	7.5n	x
MAGASINTAL	STORAGE	STORAGE	7.5n	x
VGRAD	EFF	EFFICIENCY	2n	x
NOTAT1	NOTE1	NOTE 1	70c	x
NOTAT2	NOTE2	NOTE 2	70c	x
NOTAT3	NOTE3	NOTE 3	70c	x
DEPTH	DEPTH	DEPTH	7.5n	x
STATUS	STATUS	NOTE	2c	x

### The LIT File

An example illustrating the different features of the LIT file..

The names of the features are given in Danish (Name (DK)) but in the table in addition an English name and description is included.

The column "Type" describes the format; for strings the length and a 'c', for numeric the length a dot the number of decimals and a 'n'.

Name (DK)	Name (UK)	Description	Type	Used by the GeoEditor
DGUNR	ID	BOREHOLE ID	10c	x
LBUND	LAYBOT	BOTTOM OF LAYER	5.1n	x
BJA	ROCK	ROCK	3c	
DGUSYM	SYMBOL	SYMBOL	2c	x
BLAND1	BICOM 1	BI-COMPONENT 1	2c	





BIBJA1	CON1	CONTENT OF ABOVE	3c	
BLAND2	BICOM2	BI-COMPONENT 2	2c	
BIBJA2	CON2	CONTENT OF ABOVE	3c	
BLANDK	LIMECON	LIME CONTENT	2c	
KALKH	CHALKY	CHALKY	2c	
TEXTUR	TEXTUR	TEXTUR	5c	
FARVE	COLOUR	COLOUR	5c	